Piping Systems FluidFlow

Quick Start

Owned and Copyright by Flite Software NI Ltd

Table of Contents

Foreword 0

Part I Piping Systems FluidFlow 1

1 Welcome ... 1
2 Installation ... 1
3 Activation ... 1
4 Keeping your Software Current ... 5
5 Network Issues ... 6
6 Starting the Application (Network Module) ... 9
7 Changing User Access Information (Network Module) ... 11
8 Application Layout ... 13
9 Design of a cooling water system ... 14
10 Design of a Tank Farm Gas Collection System ... 27
11 Design of a Cooling Water System. Part 2. ... 29
12 Configuration and Environment ... 32
13 Databases ... 33
14 Fluids Database ... 33
15 Database of Manually Operated Valves ... 35
16 Add a New Pump ... 38

Index 40

1 Piping Systems FluidFlow

1.1 Welcome

Welcome to Piping Systems FluidFlow a state-of-the-art fluid flow simulator. This software application allows you to simulate the flow of fluids in complex networks, taking into account the phase state of the fluid and determining heat changes. FluidFlow is more than a pipe network analysis program, it is a fully developed steady-state process-flow simulator.

System Requirements

- 1024 MB RAM (2048 MB recommended)
- Microsoft Windows Server 2008, 2003, Windows 7, Windows 8, Vista or XP
- 80MB of free hard disk space
- SVGA or higher resolution monitor (XGA recommended)
- Mouse or other pointing device

1.2 Installation

FluidFlow is supplied as a single compressed installation file - FF3SETUP.EXE. This file is available via a download from our website www.fluidflowinfo.com (preferred method).

This is a common installation file for all possible modules. Simply run the file FF3SETUP.EXE and the installer will start and take you through the setup process. You can also use the setup program to install updates into your installation folder (only executable, help files etc are updated, databases and your project files are not overwritten).

It is possible to install remotely if you are a network administrator. The installation does not require any registry entries and for users not wishing to use an installer (for example in locked environments) there is a zipped version of the application and associated folders.

This product has been fully tested and can also be installed to run under Citrix or Terminal Services.

If you intend to run many concurrent users across remote locations outside of a LAN, (i.e. a WAN across country borders) you need to purchase a global licence.

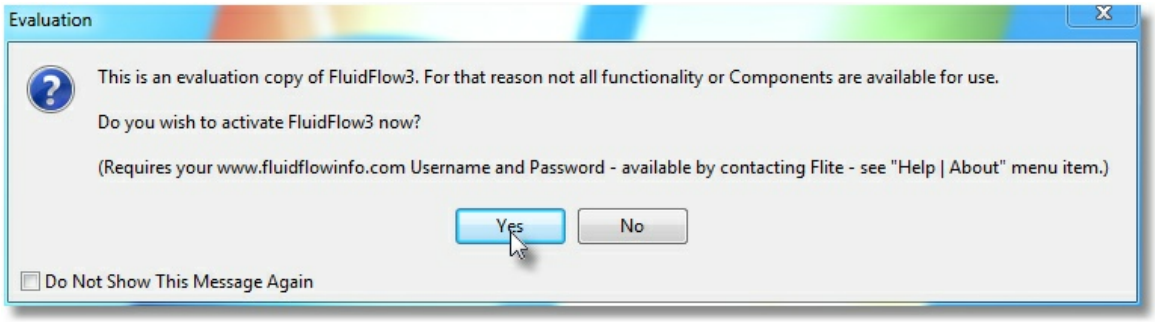
Once installed the software reverts to demo mode until it is activated. So the first thing you need to do after installation is to Activate the software. There is a simple activation process for both installed and unzipped installations.

In order to activate and run the software you **MUST** have **read/write** access to the folder where FluidFlow is installed. You cannot activate over a LAN or WAN without using terminal services, remote desktop, citrix etc, since for activation the application MUST be running in the server process workspace. For more information about the activation process see the activation chapter.

1.3 Activation

Activation is the process of configuring access to the available FluidFlow modules. After a new installation there are no active modules.

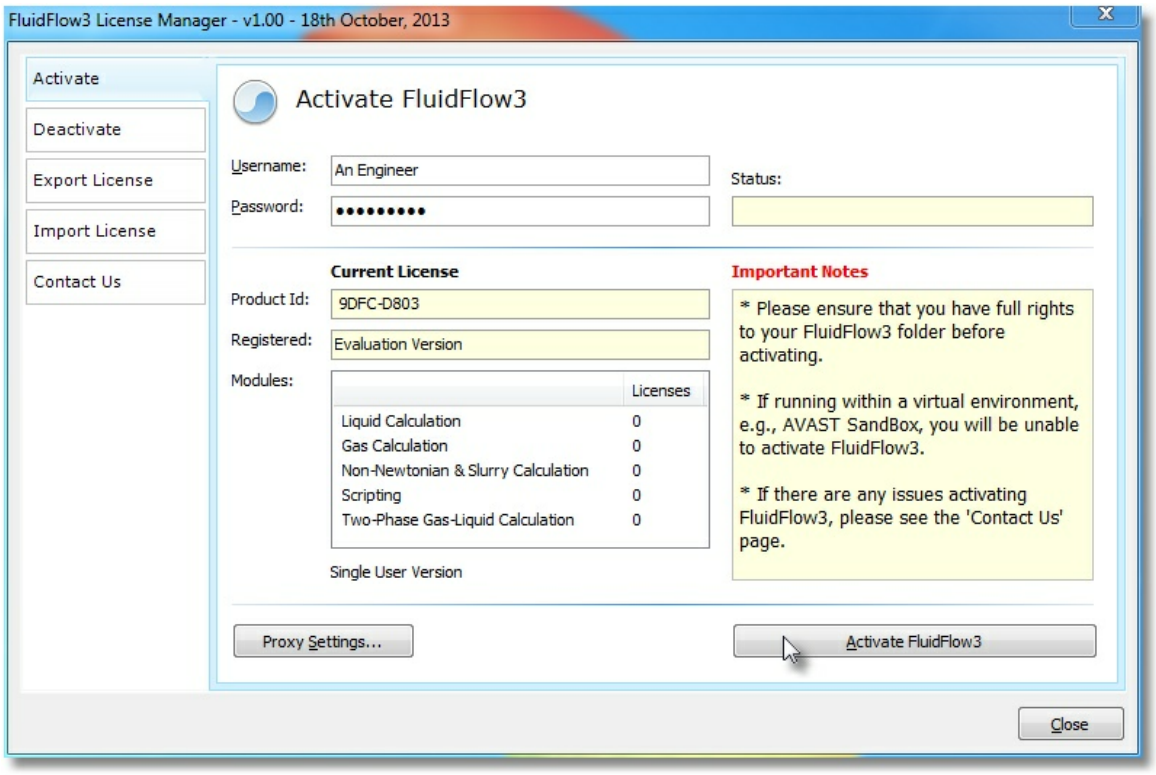
From FluidFlow V3.3, the product activation can be carried out automatically. When FluidFlow is started for the first time the following dialog appears


Evaluation
This is an evaluation copy of FluidFlow3. For that reason not all functionality or Components are available for use.
Do you wish to activate FluidFlow3 now?
(Requires your www.fluidflowinfo.com Username and Password - available by contacting Flite - see "Help | About" menu item.)
Yes
No
Do Not Show This Message Again

© <2013> ... Flite Software NI Ltd

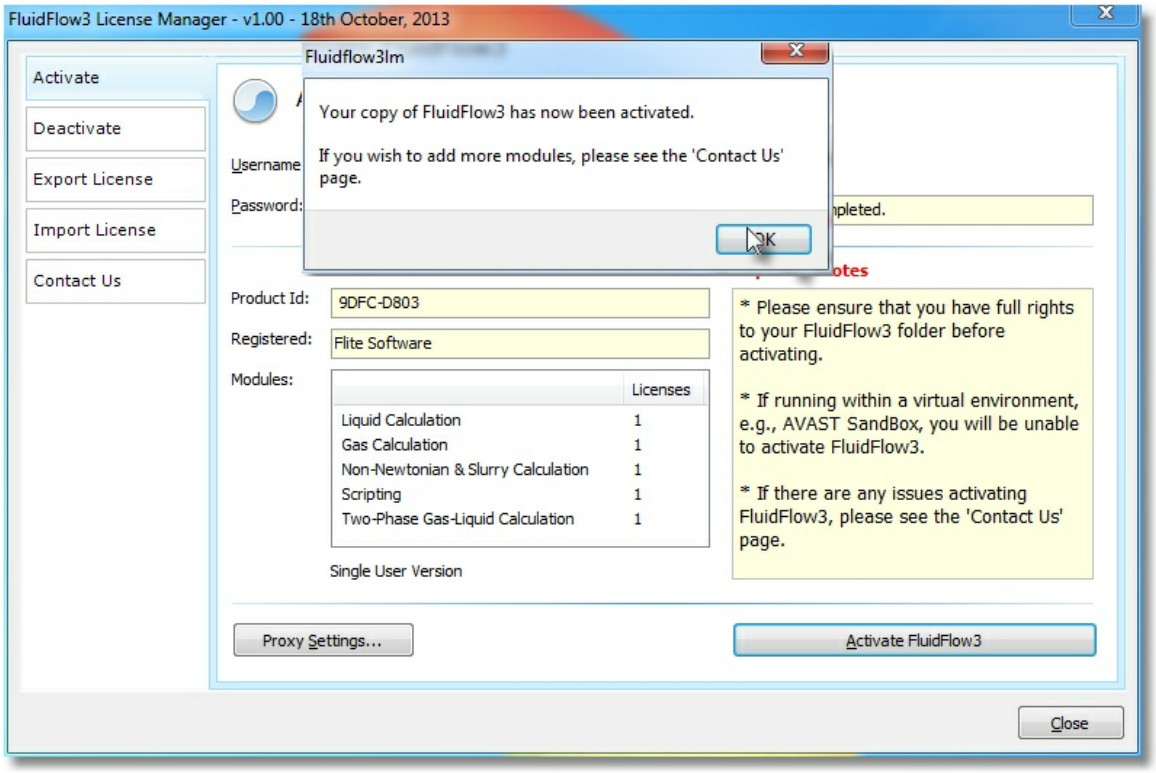
If you wish to defer activation or do not have an internet connection then press No and you can activate via email or directly from our website. If you have an internet connection then select the Yes button and the activation process will be completed as described below.

When you purchase the software or obtain a demo licence you will be provided with a Username and Password. Use the information provided in the Licence Manager as shown.



Press the Activate FluidFlow3 button and the software will automatically obtain an activation code and activate the software. If a successful activation occurs the following message appears

© <2013> ... Flite Software NI Ltd

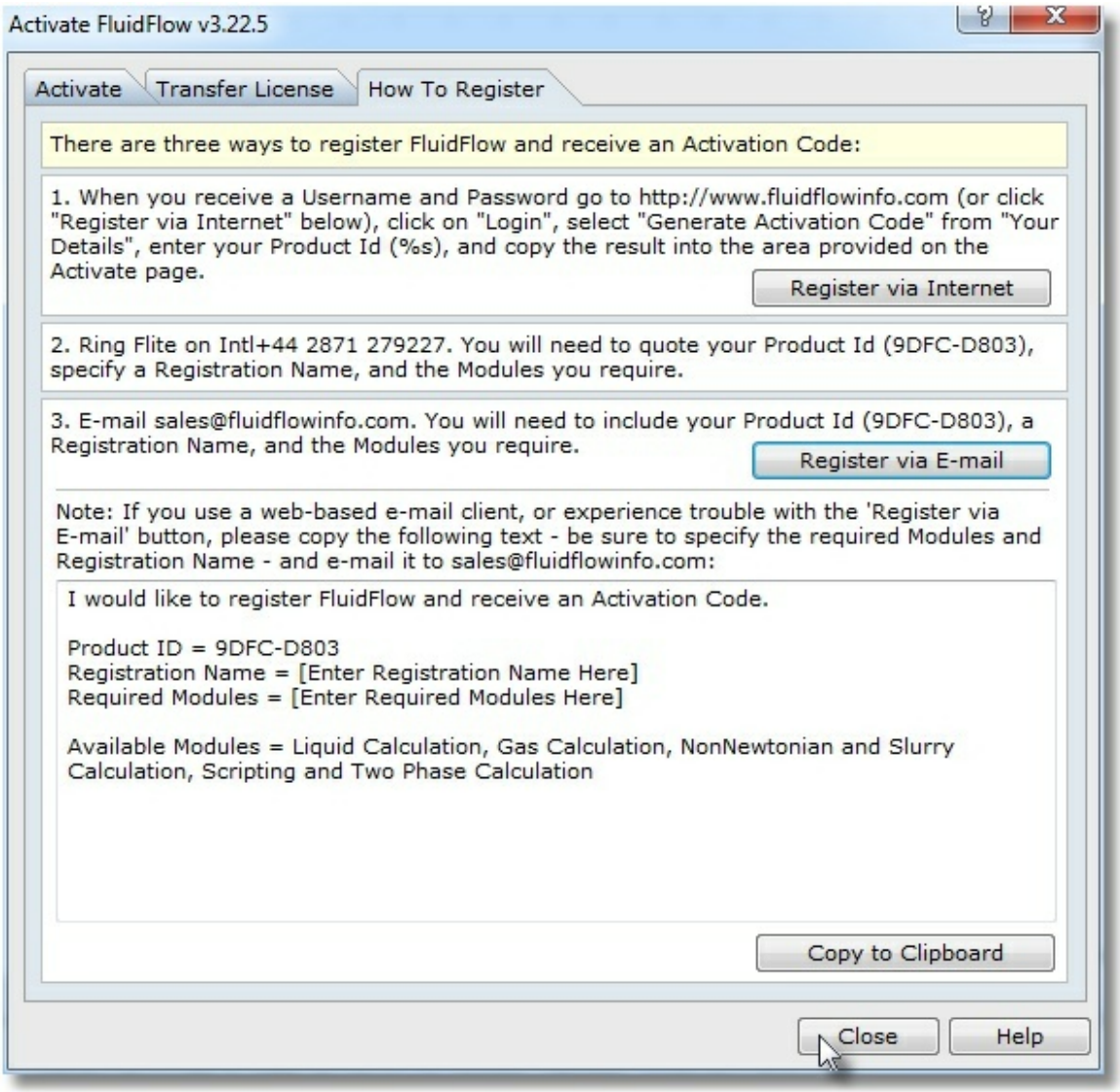

FluidFlow3 License Manager - v1.00 - 18th October, 2013
Activate
Deactivate
Export License
Import License
Contact Us
Fluidflow3Im
Your copy of FluidFlow3 has now been activated.
If you wish to add more modules, please see the 'Contact Us' page.
OK
Username
Password:
Product Id:
9DFC-D803
Registered:
Flite Software
Modules:
Licenses
Liquid Calculation 1
Gas Calculation 1
Non-Newtonian & Slurry Calculation 1
Scripting 1
Two-Phase Gas-Liquid Calculation 1
Single User Version
* Please ensure that you have full rights to your FluidFlow3 folder before activating.
* If running within a virtual environment, e.g., AVAST SandBox, you will be unable to activate FluidFlow3.
* If there are any issues activating FluidFlow3, please see the 'Contact Us' page.
Proxy Settings...
Activate FluidFlow3
Close

All of the alternative activation methods that were available prior to V3.3 are still available and these are described below.

You can use the 'Help | Activate FluidFlow...' menu option, which will display the dialog below.

Click on the "How to Register" tab and select one of the available registration methods.

© <2013> ... Flite Software NI Ltd


Activate FluidFlow v3.22.5
Activate
Transfer License
How To Register
There are three ways to register FluidFlow and receive an Activation Code:
1. When you receive a Username and Password go to http://www.fluidflowinfo.com (or click "Register via Internet" below), click on "Login", select "Generate Activation Code" from "Your Details", enter your Product Id (%s), and copy the result into the area provided on the Activate page.
Register via Internet
2. Ring Flite on Intl+44 2871 279227. You will need to quote your Product Id (9DFC-D803), specify a Registration Name, and the Modules you require.
3. E-mail sales@fluidflowinfo.com. You will need to include your Product Id (9DFC-D803), a Registration Name, and the Modules you require.
Register via E-mail
Note: If you use a web-based e-mail client, or experience trouble with the 'Register via E-mail' button, please copy the following text - be sure to specify the required Modules and Registration Name - and e-mail it to sales@fluidflowinfo.com:
I would like to register FluidFlow and receive an Activation Code.
Product ID = 9DFC-D803
Registration Name = [Enter Registration Name Here]
Required Modules = [Enter Required Modules Here]
Available Modules = Liquid Calculation, Gas Calculation, NonNewtonian and Slurry Calculation, Scripting and Two Phase Calculation
Copy to Clipboard
Close
Help

The software generates a Product Id (9DFC-D803 in the example above) directly from your computer. Simply email this Product Id together with the calculation modules you need, or have purchased, to support@fluidflowinfo.com and an activation code and registration name will be provided (usually by return).

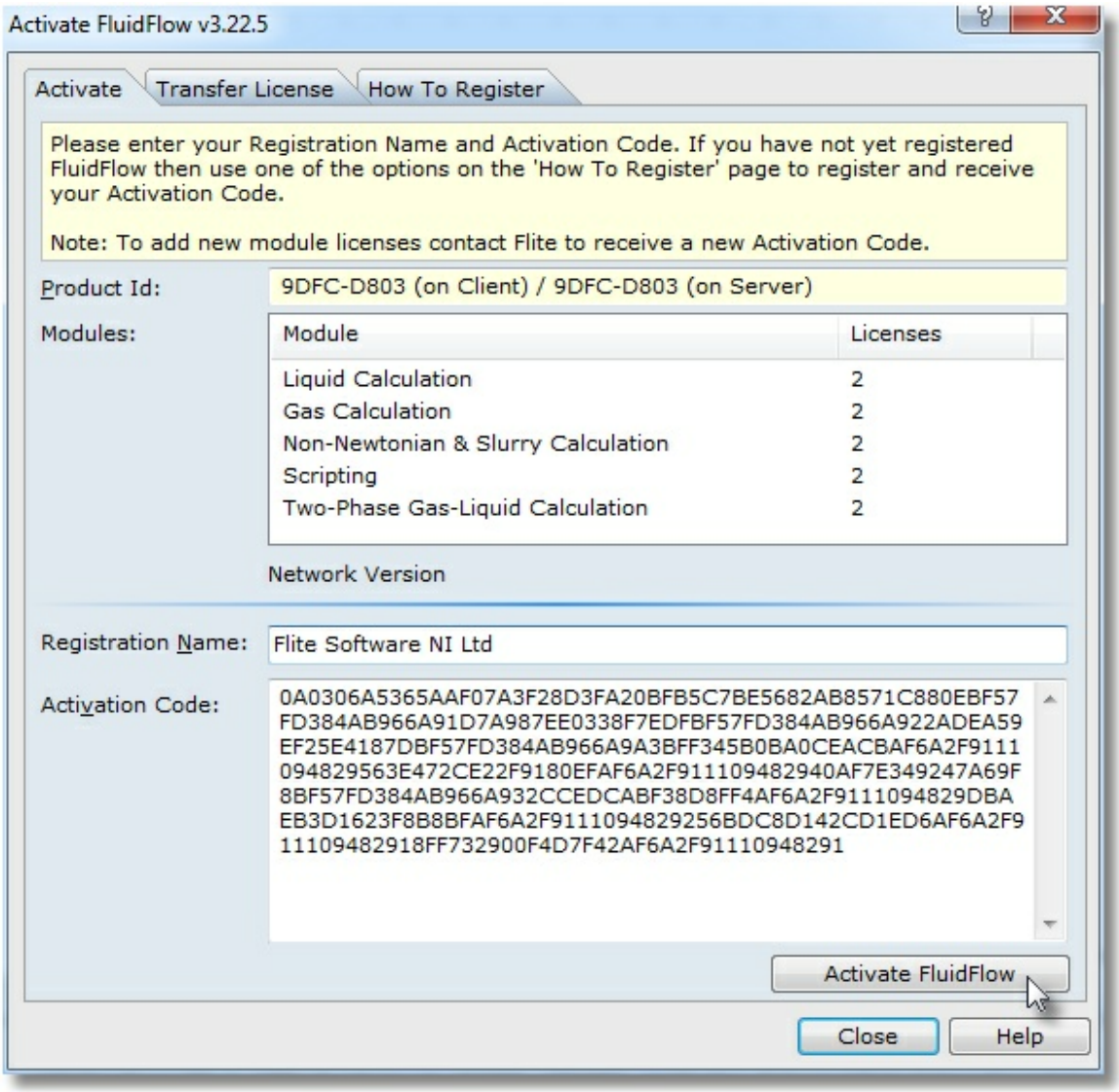
On receipt of your activation code and registration name (you can specify the registration name if you wish in the email you send to us), click on the "Activate" tab and enter the information you have received as shown below.

Assuming we have received the following activation data:

Registration Name: Flite Software NI Ltd
Activation Code:
0A0306A5365AAF07A3F28D3FA20BFB5C7BE5682AB8571C880EBF57FD384AB966A91D7A987EE0338F7EDFBF57FD384AB966A922ADEA59EF25E4187DBF57FD384AB966A9A3BFF345B0BA0CEACBAF6A2F9111094829563E472CE22F9180EFAF6A2F911109482940AF7E349247A69F8BF57FD384AB966A932CCEDCABF38D8FF4AF6A2F9111094829DBAEB3D1623F8B8BFAF6A2F9111094829256BDC8D142CD1ED6AF6A2F911109482918FF732900F4D7F42AF6A2F91110948291

Copy and paste this information from your email.

© <2013> ... Flite Software NI Ltd



Press the 'Activate FluidFlow' button and the Registration Id will now contain a code instead of Evaluation Version and the modules that you have purchased will become available.

Once activated you should quit the application and restart. This step is important for correct activation.

If you are activating a network version then you should also read the network activation and setup section for additional information.

1.4 Keeping your Software Current

Flite Software has a commitment to constant improvement of the FluidFlow product. In addition we provide an undertaking to attempt to fix bugs and annoyances in a timely manner. This commitment means that the FluidFlow product is constantly improving and so it is in your interest to stay current by using the latest release.

From Version 3 we have adopted the following numbering convention for product versions.

- All minor bug fixes for a given version will be implemented via new builds of the same version. This is in contrast to the old system used for V2 product, which always incremented the version number for all bug fixes.

- Only enhancements or new features will cause a version number increment to occur. For example, in the past any new bugs reported for version 2.26 would be fixed and appear in a 2.27 version only. Now, any new bugs reported for version 3.15 Build 1 will be fixed and will appear in version

© <2013> ... Flite Software NI Ltd

3.15 Build 2. Written as 3.15.2

- All version numbers have the format [Major Version].[Minor Version] Build Number, for example, 3.22 Build 5 means a Version 3 product with a Minor Version number of 22 and a Build Number of 5. Written as 3.22.5

It is your responsibility to keep your software current via our website. We have a Software Assurance Policy that enables continued access to the downloads area of our website:

http://www.fluidflowinfo.com/Downloads/Downloads.asp

1.5 Network Issues

Note: This section is only relevant if you have the Network module.

Network Installation

The installation process on the network is the same as stand-alone: simply run FF3SETUP.EXE. FluidFlow can be installed across a network as long as you use terminal services, remote desktop, or a similar access method. The network folder MUST have write access, which usually means you must have admin rights during the installation process. FluidFlow does not use or require access to the Registry.

After installation your network users **must** be given permission/access rights to the FluidFlow folder. We would suggest that users are given full access to the folder that FluidFlow is installed in, e.g., \Flite\FluidFlow3, and that this access is propagated through the sub-folders.

However, if you wish to restrict full access, FluidFlow requires read access to all folders, and additionally, write access to the following folders:

- \Flite\FluidFlow3
- \Flite\FluidFlow3\Data (and all sub-folders)
- \Flite\FluidFlow3\Preferences (and all sub-folders)

Note: In Windows XP, it is not enough to just change the properties of the folder(s), you must also check the 'Allow network users to change my files' checkbox in the Network Sharing and Security Properties for the folder. (Note: This can be accessed in Windows Explorer via the Properties context menu item, then the Sharing tab.)

Overriding the location of the Read-Write Folders

With some network installations, Administrators prefer to have control over which folders client users have read-write access to. To cater for this, FluidFlow provides PSFF.INI which allows the Administrator to specify the default location for \Data, \Preferences, and the Network Access files.

Note: FluidFlow does not create this INI file itself; it is the responsibility of the Network Administrator to create PSFF.INI and place it in the same folder as PSFF.EXE. If you wish to use a psff.ini file follow the explanations below;

```
PSFF.INI
[Options]
; Data
UseCommonAppDataFolder=0
DataFolder=

; Preferences
UseLocalAppDataPreferencesFolder=0
PreferencesFolder=

; Network Access
UseCommonAppDataNetworkAccessFolder=0
NetworkAccessFolder=
```

Data

If UseCommonAppDataFolder=1 then FluidFlow will use the "C:\Document and Settings\All

© <2013> ... Flite Software NI Ltd

Users\Application Data\FluidFlow3\Data" folder. Otherwise FluidFlow will use the value of the DataFolder entry. If this is blank, or PSFF.INI does not exist, then FluidFlow will use "\FluidFlow3 \Data".

NOTE: If DataFolder is specified then this folder MUST exist.

Preferences

If UseLocalAppDataPreferencesFolder=1 then FluidFlow will use the "C:\Document and Settings\[Username]\Local Settings\Application Data\FluidFlow3\Preferences" folder. Otherwise FluidFlow will use the value of the PreferencesFolder entry + [Username]. If this is blank, or PSFF.INI does not exist, then FluidFlow will use "\FluidFlow3\Preferences\[Username]". [Username] is the windows log on name of the user.

Network Access

If UseCommonAppDataNetworkAccessFolder=1 then FluidFlow will use the "C:\Document and Settings\All Users\Application Data\FluidFlow3" folder. Otherwise FluidFlow will use the value of the NetworkAccessFolder entry. If this is blank, or PSFF.INI does not exist, then FluidFlow will use "\FluidFlow3".

NOTE: If NetworkAccessFolder is specified then this folder MUST exist.

Network Activation

Although FluidFlow can be installed across a network, it **must** be activated from the installation on the server machine. This means you must be either physically at the server or accessing the server remotely via terminal services, citrix etc. You cannot activate the server via normal LAN access. When contacting Flite for an Activation Code, please state how many users the network license is for.

To contact Flite and receive your Activation Code:

- Ring Flite on Int+44 2871 279227. You will need to quote your Product Id, specify a Registration Name, and the Modules you require.
- E-mail salest@fluidflowinfo.com. You will need to include your Product Id, a Registration Name, and the Modules you require.

When you enter the Activation Code you will need to restart FluidFlow for it to take effect.

Client Setup

After installing and activating FluidFlow on the network server you can set up the client machine by placing a link on the client's desktop to FluidFlow. To do this, assuming that FluidFlow3 is the name of the shared folder:

1. Right-click on the desktop and select the 'New | Shortcut' menu item.

2. Type the location of the PSFF.EXE file, e.g., '\\Server\FluidFlow3\PSFF.exe' and click 'Next'.

3. Enter the title for the shortcut, e.g., 'FluidFlow3 (Network)' and click 'Finish'.

Note: If you do place a link on the desktop to FluidFlow on the network server, then this may affect the startup speed of the client machine. This is because on startup the client will search the Network Neighbourhood to find the link's target. Normally, if the network server is always on, this will take no noticeable time; however, if the network server is down, you may notice some small extra delay in startup. (Note: This is not specific to FluidFlow, but is true of all applications linked to a network resource.)

Help Files on the Network

From FluidFlow release V3.22.4 help files work across a LAN or WAN automatically.

If you are not using the current production release then the following info may be important in order

© <2013> ... Flite Software NI Ltd

to get your help files operating over a network.

Recent security updates to Windows XP have introduced some severe restrictions for accessing HTML Help files across network drives. Under Windows XP and Vista most file links in HTML Help files will now generally not work at all and HTML Help itself is also severely restricted. Without registry changes on the user's computer HTML Help now cannot be used at all on networks. This has an effect on the FluidFlow Network Version as all client machines that try to display the Help file will receive a "Page not found" message.
More details and a fix for this problem are available on the Help & Manual website.
Alternatively, there is a workaround to allow a client machine to display Network-based HTML Help files, but it does involve modifying the Registry on the client machine. To do this:

1. Click 'Start', click 'Run', type 'regedit', and then click 'OK'.

2. Locate and then click the following subkey:
HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\HTMLHelp\1.x\ItssRestrictions
Note: If this registry subkey does not exist, create it. To do this, follow these steps:
a. On the 'Edit' menu, point to 'New', and then click 'Key'.
b. Type 'ItssRestrictions', and then press 'ENTER'.

4. Right-click the ItssRestrictions subkey, point to 'New', and then click 'DWORD Value'.

5. Type 'MaxAllowedZone', and then press 'ENTER'.

6. Right-click the MaxAllowedZone value, and then click 'Modify'.

7. In the Value data box, type '1', and then click 'OK'.

This will allow you to access CHM files on a shared network folder. For more information see: http://support.microsoft.com/?kbid=896054

Note: Always make a backup of the Registry before making any modifications so that you can 'rollback' the changes if anything goes awry. To do this, run Regedit, select the 'File | Export' menu item, select the 'All' option from 'Export Range', enter a filename and click 'Save'. You can later use the 'File | Import' menu item if you want to revert to your original Registry.

Troubleshooting

The most usual issues reported for network installations and their resolutions are given below...

1. Unable to Activate
This is because you are trying to activate from a client machine. This is not possible because you are running the application in the client workspace and not the server workspace. For the purpose of activation **ONLY,** you can overcome this issue by connecting to the server via remote desktop (terminal services, citrix etc) or being physically present at the server to activate. So to activate a network version, run the application via remote desktop, or be physically at the server.

Another possible reason for "unable to activate" is because you do not have the correct read/write/modify permissions to the folder (and all sub folders) where FluidFlow is installed.

2. "User Limit [1] reached. No more users allowed." message always displayed no matter how many licenses. This is caused by the incorrect sharing of the FluidFlow folder. For more information see the Network Installation section above.

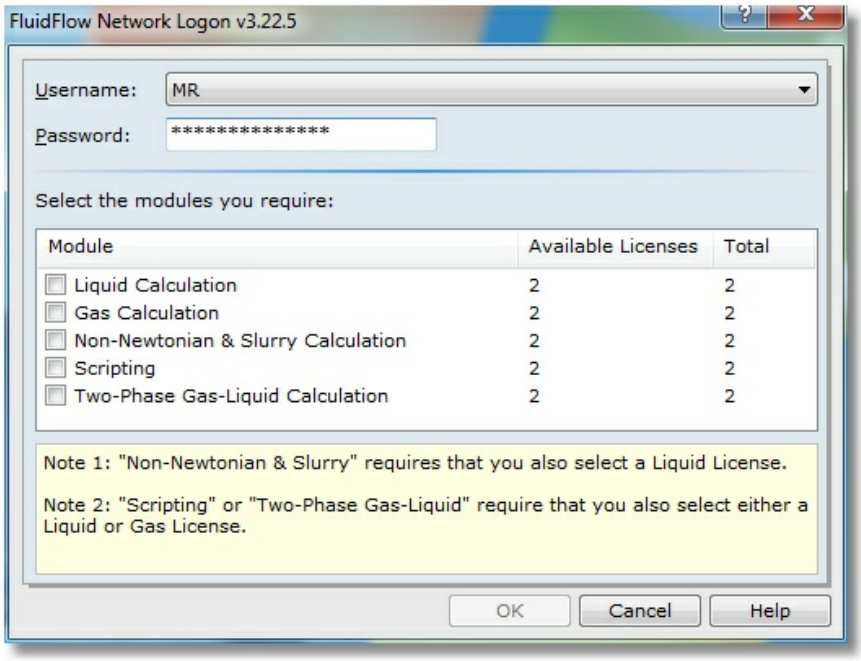
© <2013> ... Flite Software NI Ltd

1.6 Starting the Application (Network Module)

If you have the network module activated the product starts slightly differently to the start-up of the stand-alone version.

If you do not have the network module activated then skip this section.

The first step in the start-up for a network user is a simple logon screen:



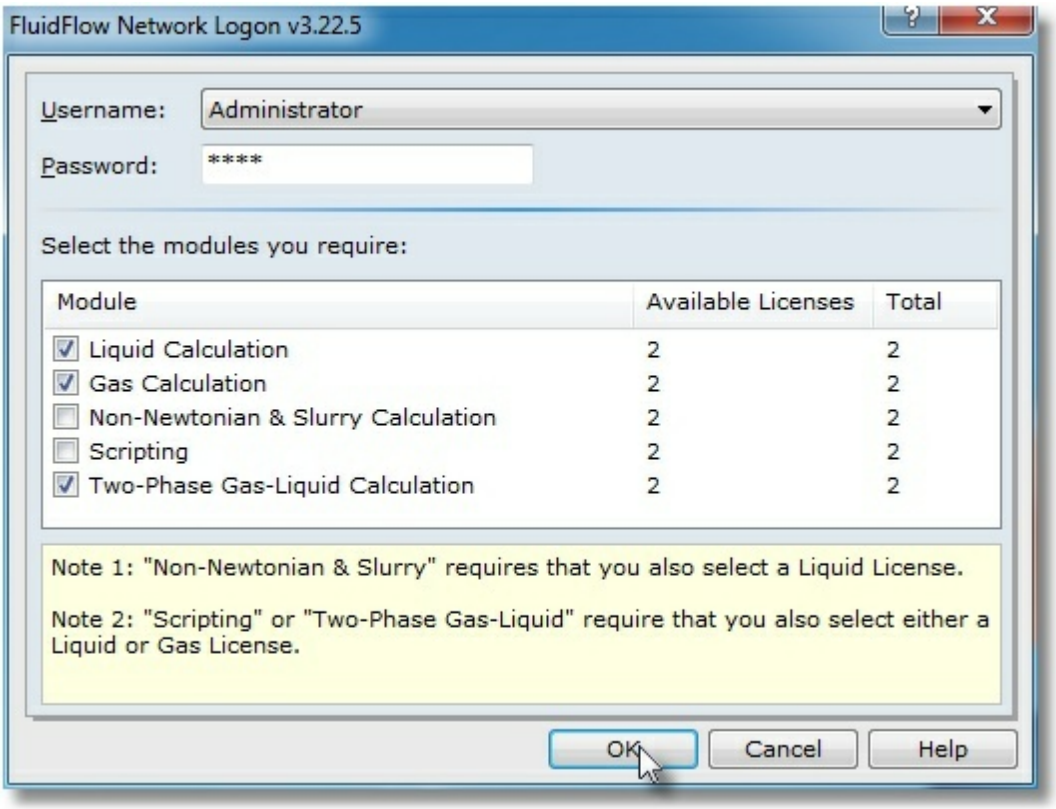
Select your username from the list and enter the password given to you by the system administrator.

Select from the available licenses the calculation functionality you will need. In the above below we have selected liquid, gas and Two-Phase calculations.

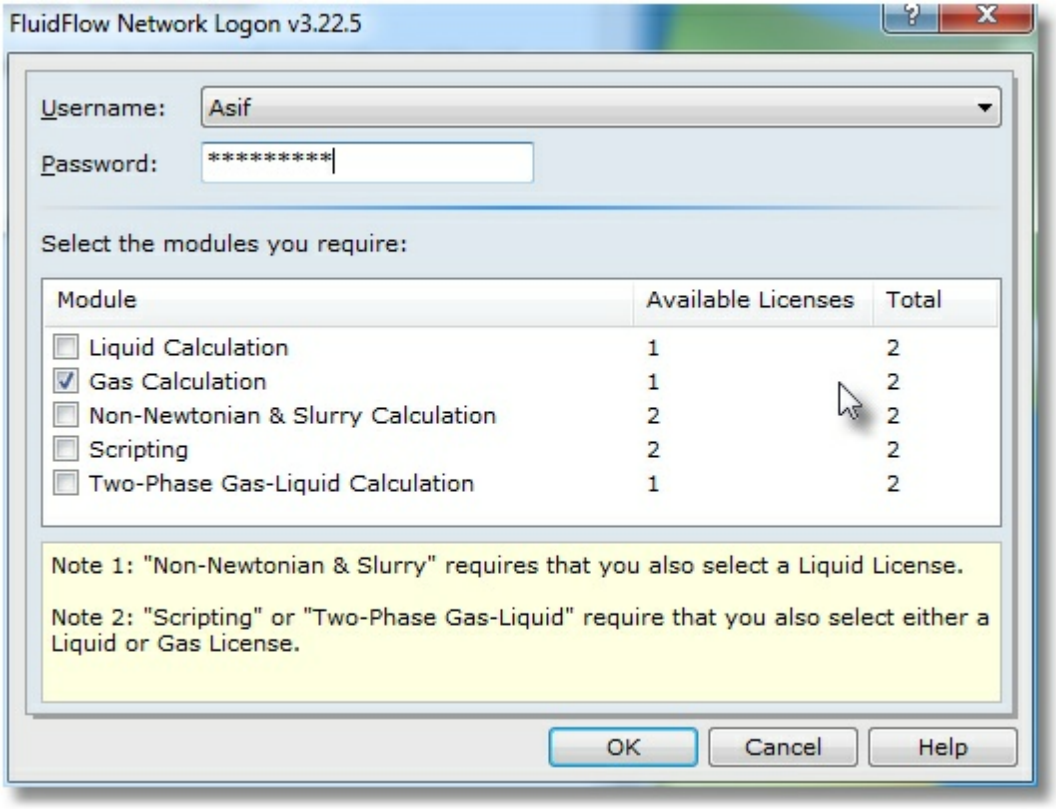
Press the OK button and the application will start up proper.

In the below example the user Administrator will be using 3 modules from a total pool of 5.

© <2013> ... Flite Software NI Ltd


FluidFlow Network Logon v3.22.5
Username: Administrator
Password: ****
Select the modules you require:
Module
Available Licenses
Total
Liquid Calculation 2 2
Gas Calculation 2 2
Non-Newtonian & Slurry Calculation 2 2
Scripting 2 2
Two-Phase Gas-Liquid Calculation 2 2
Note 1: "Non-Newtonian & Slurry" requires that you also select a Liquid License.
Note 2: "Scripting" or "Two-Phase Gas-Liquid" require that you also select either a Liquid or Gas License.
OK
Cancel
Help

The next user to log on will see the following Logon dialog.


FluidFlow Network Logon v3.22.5
Username: Asif
Password: *********
Select the modules you require:
Module
Available Licenses
Total
Liquid Calculation 1 2
Gas Calculation 1 2
Non-Newtonian & Slurry Calculation 2 2
Scripting 2 2
Two-Phase Gas-Liquid Calculation 1 2
Note 1: "Non-Newtonian & Slurry" requires that you also select a Liquid License.
Note 2: "Scripting" or "Two-Phase Gas-Liquid" require that you also select either a Liquid or Gas License.
OK
Cancel
Help

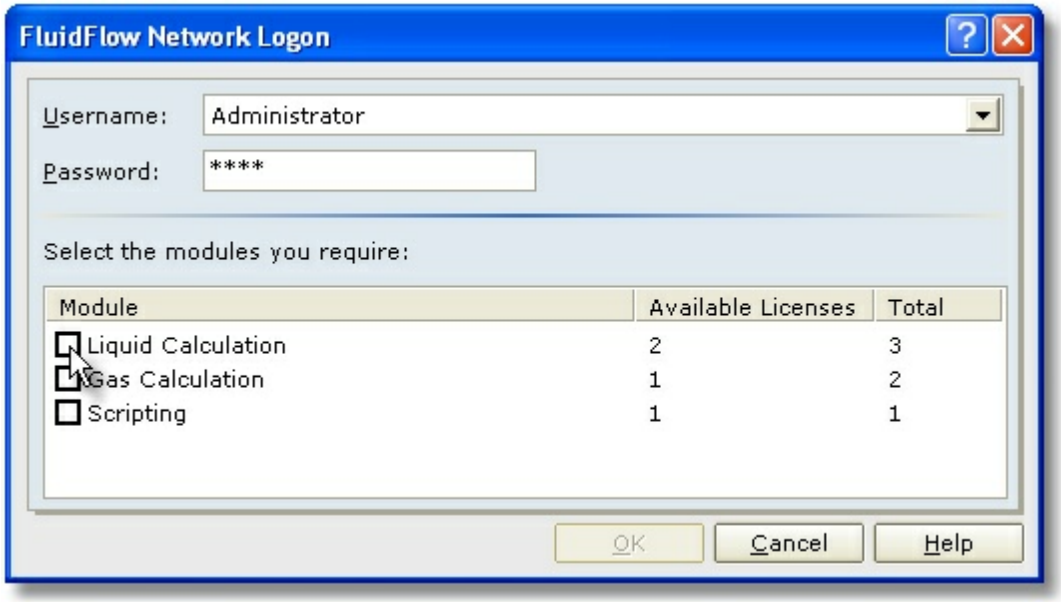
Notice that there are now less licence's available as liquid, gas and Two-Phase modules were taken by Administrator who logged on first.

You can skip to the next section unless you are the administrator and wish to set up a group of users, delete a user, or change a password.

© <2013> ... Flite Software NI Ltd

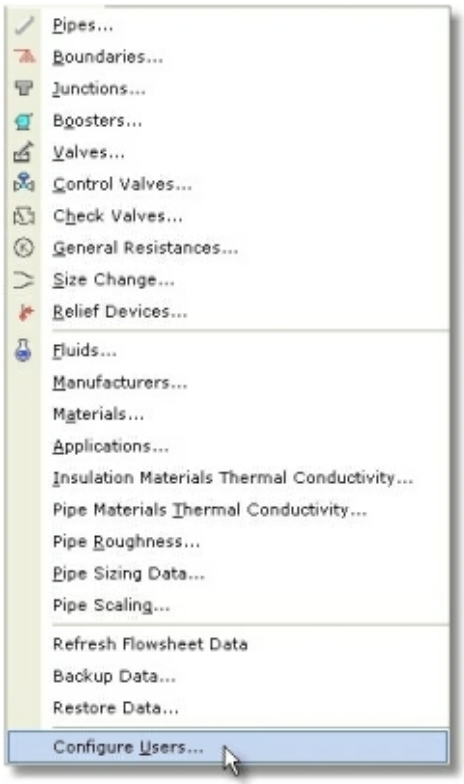
1.7 Changing User Access Information (Network Module)

To be able to make changes to user information you must logon to the application as Administrator. The default administrator password is PSFF. Logon as Administrator as shown below:



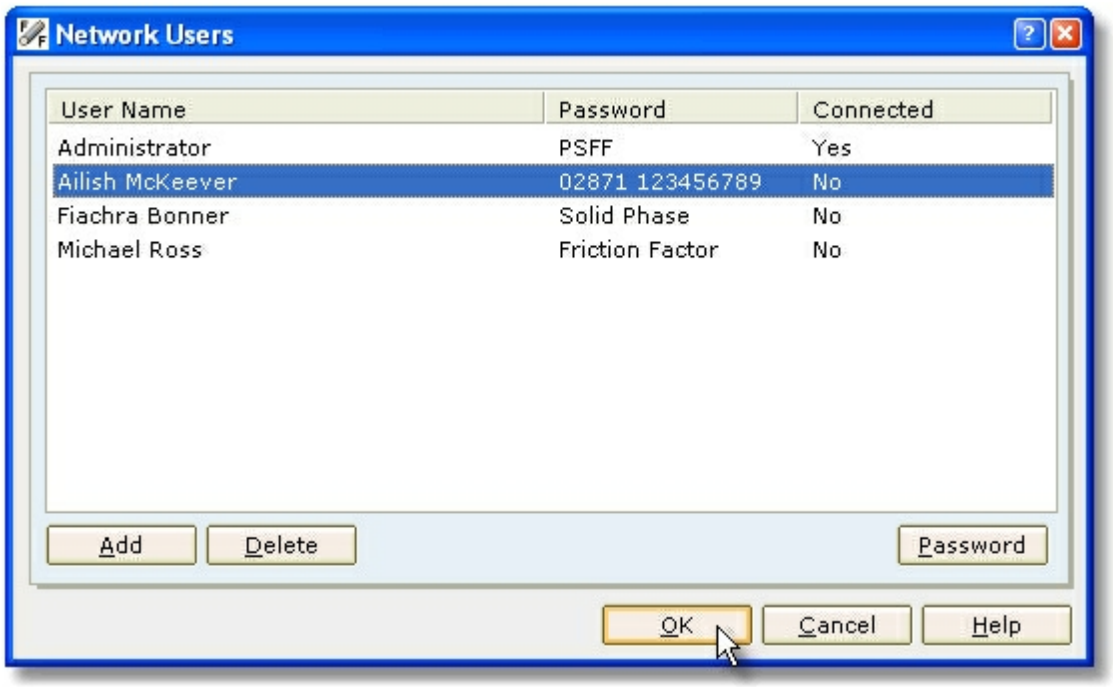
You must select at least one module in order to log on.

The application will start in the normal manner but an additional item will appear at the end of the 'Database' menu items as shown below:



Select the 'Configure Users...' menu option to obtain the dialog from which all adjustments can be made.

© <2013> ... Flite Software NI Ltd



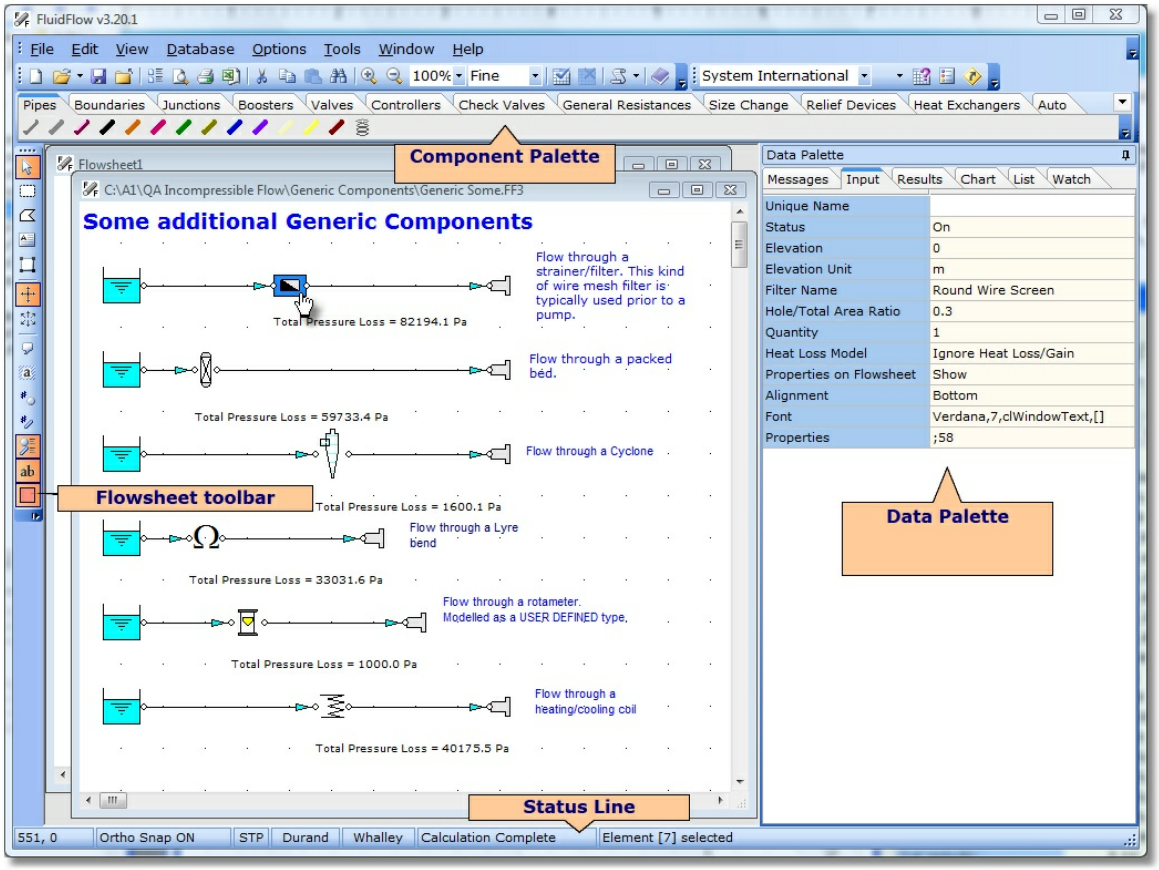
From this dialog you can add or delete users, change passwords, or just view the current connections.

It is not possible to delete the Administrator.

© <2013> ... Flite Software NI Ltd

1.8 Application Layout

The application work screen consists of two main windows. The flowsheet window where a schematic layout of the piping network is developed or built, and the data palette where data input is made, results and warnings are shown, etc.
You can have as many flowsheets open as desired. From a flowsheet viewpoint the application behaves similar to Microsoft Word's Multiple Document Interface, that is, you can tile, cascade, and select a flowsheet from the window menu or by clicking on the caption bar of the window. If you double-click the caption bar the flowsheet fills the available work area.
The data palette size can be adjusted by dragging the left side of the window border. The data palette is synchronized to the flowsheet, so if you click on a flowsheet element, the data palette is refreshed to reflect the current selections(s). This process also works in reverse, for example, if you click an element warning in the data palette, or a list item, the flowsheet selection updates to reflect this.


FluidFlow v3.20.1
File Edit View Database Options Tools Window Help
Component Palette
Flowsheet1
C:\A1\QA Incompressible Flow\Generic Components\Generic Some.FF3
Some additional Generic Components
Flow through a strainer/filter. This kind of wire mesh filter is typically used prior to a pump.
Total Pressure Loss = 82194.1 Pa
Flow through a packed bed.
Total Pressure Loss = 59733.4 Pa
Flow through a Cyclone
Flowsheet toolbar
Total Pressure Loss = 1600.1 Pa
Flow through a Lyre bend
Total Pressure Loss = 33031.6 Pa
Flow through a rotameter. Modelled as a USER DEFINED type.
Total Pressure Loss = 1000.0 Pa
Flow through a heating/cooling coil
Total Pressure Loss = 40175.5 Pa
Status Line
Data Palette
Messages Input Results Chart List Watch
Unique Name
Status On
Elevation 0
Elevation Unit m
Filter Name Round Wire Screen
Hole/Total Area Ratio 0.3
Quantity 1
Heat Loss Model Ignore Heat Loss/Gain
Properties on Flowsheet Show
Alignment Bottom
Font Verdana,7,clWindowText,[]
Properties ;58
Data Palette
551, 0
Ortho Snap ON
STP
Durand
Whalley
Calculation Complete
Element [7] selected

At the top of the work screen are three rows of operators: (1) a set of drop down menus; (2) a toolbar; (3) the component palette. The component palette consists of a series of tab options. Within each tab are the equipment item icons or elements that are used to build a fluid network.
A flowsheet toolbar is positioned along the left hand side of the screen. Options here determine how you access and utilize the flowsheet window.

© <2013> ... Flite Software NI Ltd

1.9 Design of a cooling water system

This section is based on a simple example that will illustrate some of the basic concepts you can use in FluidFlow. The example involves designing a cooling water distribution system to three heat exchangers.

The topics covered in this example design are:

- Flowsheet and model building basics
- How to enter data
- How to interpret results
- How to make changes to the model so that we can obtain a better design

Problem Statement:

It is desired to provide a balanced cooling water flow from a cooling tower to 3 shell and tube heat exchangers HE1, HE2 and HE3. The size of the heat exchangers has already been determined from the process requirement and is summarized in the following table.

Table 1

Name	Heat Load in W	Tube length in m	Tube diameter in m	Number of tubes
HE1	200000	3	0.012	130
HE2	250000	3	0.012	165
HE3	170000	3	0.012	110

The cooling water is to flow through the tubes and the maximum inlet summer temperature of the cooling water will be 25°C. The design temperature rise of the cooling water is 10°C. The elevation of the exchangers above the pump centerline is 3m and the exchangers are approximately 7 m apart. Each exchanger has 2 tube passes.

We need to design/specify the following items.

- Pipe sizes to be used.
- The method we will use to balance the flow through each exchanger.
- How to make a pump selection.
- We need to consider what happens to the exit cooling water temperature of HE2 if the heat load is increased by 33%.

The completed example can be found in the Examples folder "Cooling Water Distribution. First Design Iteration", "Cooling Water Distribution. Second Design Iteration" ... through to "Cooling Water Distribution. Final Design Iteration"

Building the model in the FluidFlow flowsheet:

Before we start building the model, let us consider a moment, the design approach we intend to use. If you have the autoselection or scripting modules installed you would almost certainly use a different approach to that taken in this example.

Without these modules we will need to take more of a trial and error approach.

With the design of all systems the initial question we need to answer is where do we start and end the model i.e. where do we and how do we define the model boundaries. For this design we will start the model at the cooling tower sump and end the model at the top of the cooling tower. This means the cooling tower will not be included in this model. Most cooling water systems have a supply header taking fresh cooling water to each individual exchanger and a collection return header, we will use this same approach.

Finally, before we start building the model we need to consider the cooling water flow we need to each exchanger branch. The flow to each exchanger is determined by a heat balance. The heat transferred to the cooling water will be:

Heat Transferred (W) = mass flow (kg/s) x specific heat capacity (J/kg) x temperature rise (°C)

The specific heat of water at 30° C is approx 4154 J/kg, so from Table 1 we see that the mass flow needed to HE1 will be 200000 / (4154 x 10) = 4.81 kg/s. Summarizing in Table 2

Table 2

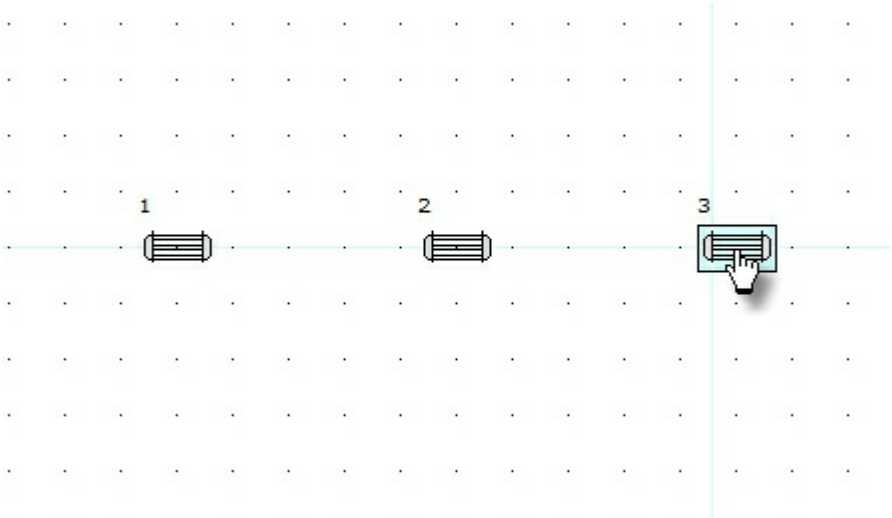
Heat Exchanger Name	Mass Flow in kg/s

© <2013> ... Flite Software NI Ltd

HE1	4.81
HE2	6.02
HE3	4.09

We will start building the model by placing 3 shell and tube exchangers onto the flowsheet. Select the shell and tube exchanger by clicking on the component palette Heat Exchangers Tab and by selecting the shell and tube icon.

Drop 3 heat exchangers onto a new flowsheet as shown below:



As we drop each element (or component) onto the flowsheet, default data is associated with the element. The default data for each element can be seen in the data palette by clicking on the Input Tab. Often we need to change some value(s) in the default data to meet our needs. For now we will continue building and come back later to change each individual element as necessary.

The reason we are deferring this task is that there are many group features built into FluidFlow to aid data editing and setup, which we can use later.

Next we will add the two boundaries.

Inlet Boundary:
For the cooling water inlet boundary we need a boundary that can represent the cooling water sump. We know that the sump is open to atmosphere and that during normal operation the liquid level in the sump is 0.5m above the pump centreline.

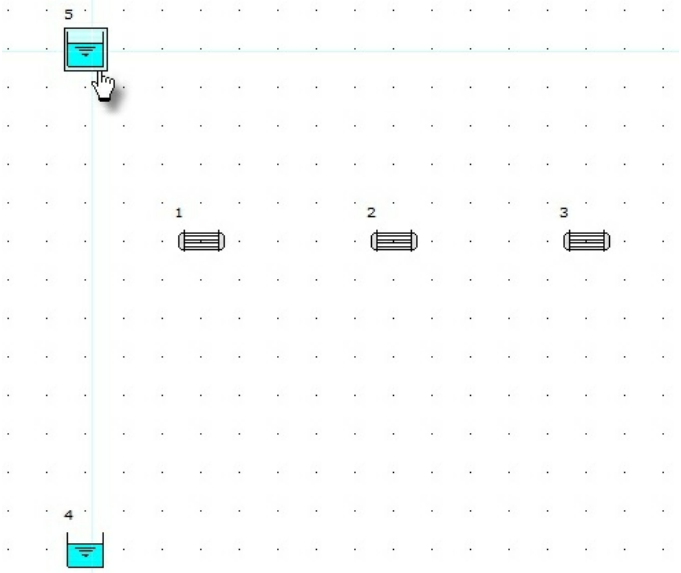
If we specify the pressure at any boundary then FluidFlow will calculate the flow that will be delivered to the system. In our design we know the design flow that is needed, because this is determined by the heat load of the exchangers. Later we will make a pump selection that will provide us with the correct flow.

Select the known flow boundary by clicking on the component palette Boundaries Tab and by selecting the known pressure icon. Place the known pressure element on the flowsheet anywhere below the 3 heat exchangers.

Outlet Boundary:
The collection return line eventually leads back to a cooling tower. At this boundary we know (or it is a design specified condition) the pressure that we must be above in order for the system to work. The water pressure necessary at the exit boundary is the sum of the elevation we need to rise to the top of the cooling tower + any pressure required to overcome the loss in the flow distribution system feeding the cooling water tower. The elevation of the cooling tower inlet above the pump centerline is 6m and the manufacturer of the cooling tower requires a minimum pressure loss of 30000 Pascals for the flow distribution to work effectively. We will therefore select a known pressure element for the exit boundary.

© <2013> ... Flite Software NI Ltd

At this point the flowsheet should look something like;.



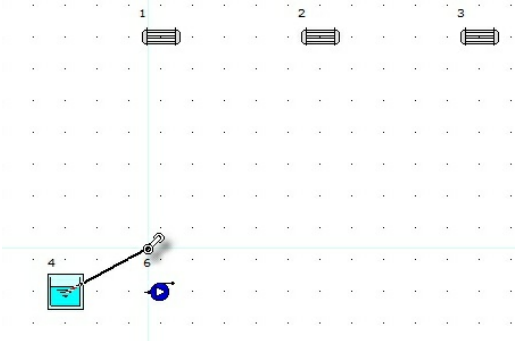
To add a pump with a known flow (in this case our design flow of 14.92 kg/s), we need to select the pump element found on the Auto tab. If we select a pump from the boosters tab this will represent a specific pump and the flow we obtain in the system will be that dictated by the intersection of the pump and system curves. Chances are that the specific pump selection will be incorrect, so we will defer the selection of the actual pump model until we have sized the pipes. To do this we use the pump from the Auto tab and specify the system design flow. The auto pump element will calculate the head required to deliver our design flow and thus provide us with the information to make the correct pump model selection.

Place an auto pump on the flowsheet to the right of node 4 (the node representing the cooling tower sump).

We are ready to start connecting pipes. FluidFlow makes pipe connecting very easy, because there is no need to include bends. These are added for you as you draw the pipes. We will again defer the task of editing data values as we connect pipes. Right now we are only concerned with building the model connectivity.

Click the pipe tool on the steel pipe icon.

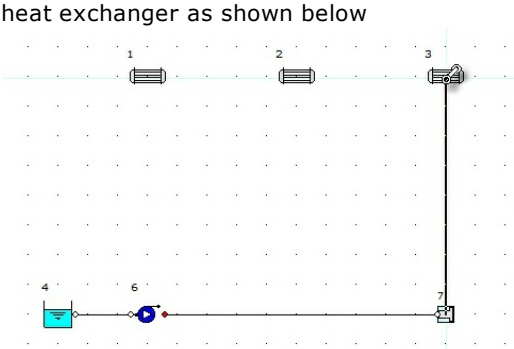
As you move the cursor over the flowsheet the shape changes to a pipe .
Click on the known pressure boundary and then move the mouse to be directly over the auto pump element, then click the left mouse button.



FluidFlow will then complete the pipe connection from the cooling water sump to the pump. While the

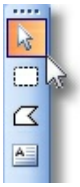
© <2013> ... Flite Software NI Ltd

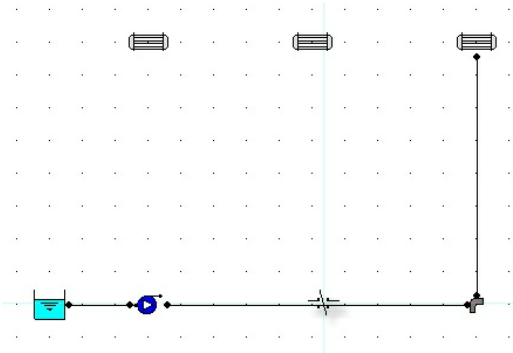
cursor is over the pump make a second left mouse click and then move the mouse cursor to the right beneath heat exchanger 3 and make another left mouse click. A pipe is created starting at the pump and terminating at an open end. Click on the open end and move the cursor to lie over the heat exchanger as shown below



Click on the exchanger, to complete the connection from the open end to the far right heat exchanger. Notice that the open pipe has changed to a bend.

f you make a mistake, click on the selector icon, in the flowsheet toolbar, select the wrongly connected element and use the Ctrl and Delete keys together to delete the selected element

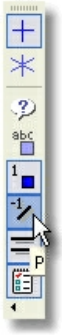




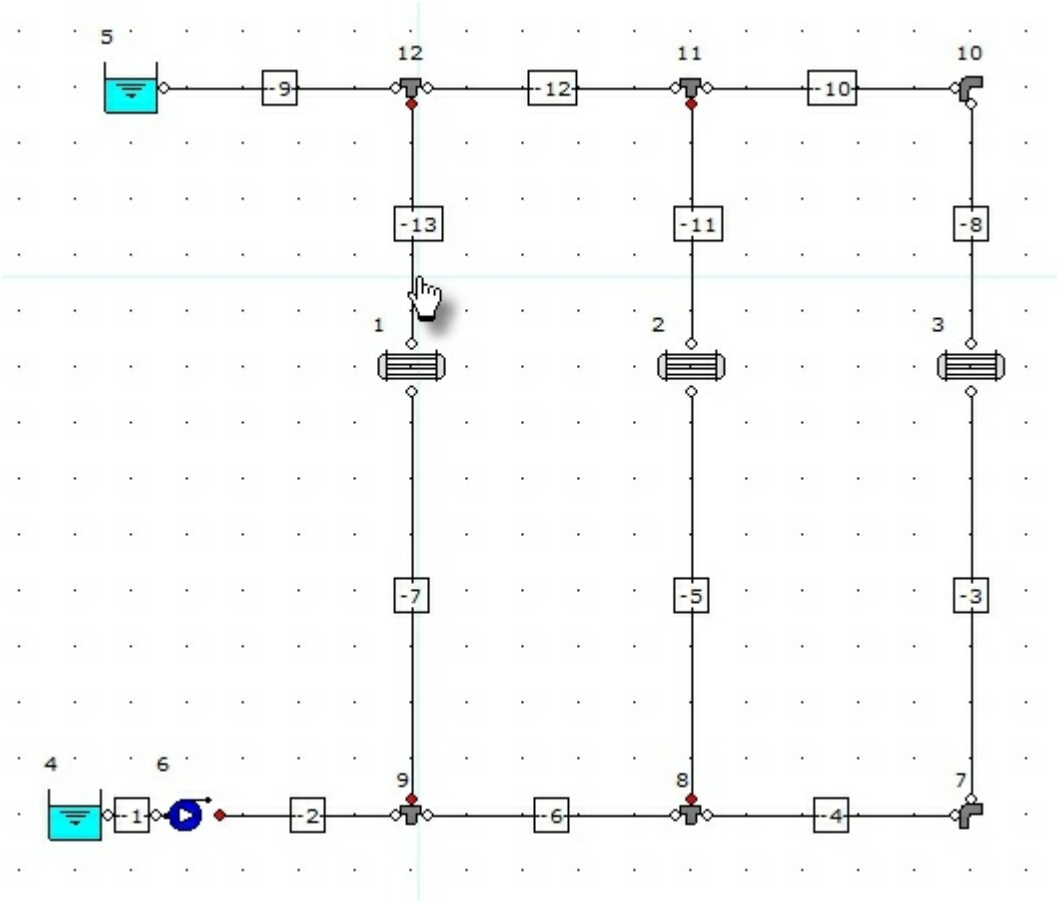
Move the cursor to the location shown above. Notice how the cursor changes from a pipe to a split pipe as we move over a pipe that can be split. If we click here the pipe will be split and we can make the connection to the middle heat exchanger.

The split pipe has converted itself into a Tee connection. This type of junction, because it is made dynamically, adjusts itself depending on the number of pipes connected. For example a single pipe connected and the junction is an open ended pipe, two connected pipes and the junction transforms to a bend, three connected pipes the junction becomes a tee or wye and with four connected pipes the junction becomes a cross.

Make further connections so that we end up with a connected network as shown next.

Turn on the pipe numbering from the flowsheet toolbar.  Note that pipe numbers go from -1 ... -n and that other elements (nodes and text) are numbered 1 ... n.

© <2013> ... Flite Software NI Ltd



Changing the default data using the flowsheet and data palette:

Up until this point no data entry has been made, we have focused on describing the element connectivity. This means that each element will have default data values according to the current environment set in use when the element was placed on the flowsheet (see Customizations and Environment section for more information about environment sets).

You can select any element on the flowsheet at any time by clicking on the element, after first using (clicking on) the selector icon.

First we need to enter all the pipe lengths.
Table 3 shows the pipe lengths that are fixed by the physical plant layout and also the number of bends in each pipe section.

Table 3

Pipe Number	Pipe Length in m	Number of 90° bends in pipe
-1	5	2
-2	12	3
-3	3	2
-4	8	0
-5	3	2
-6	8	0
-7	3	2
-8	3	2
-9	18	5
-10	8	0
-11	3	2
-12	8	0
-13	3	2

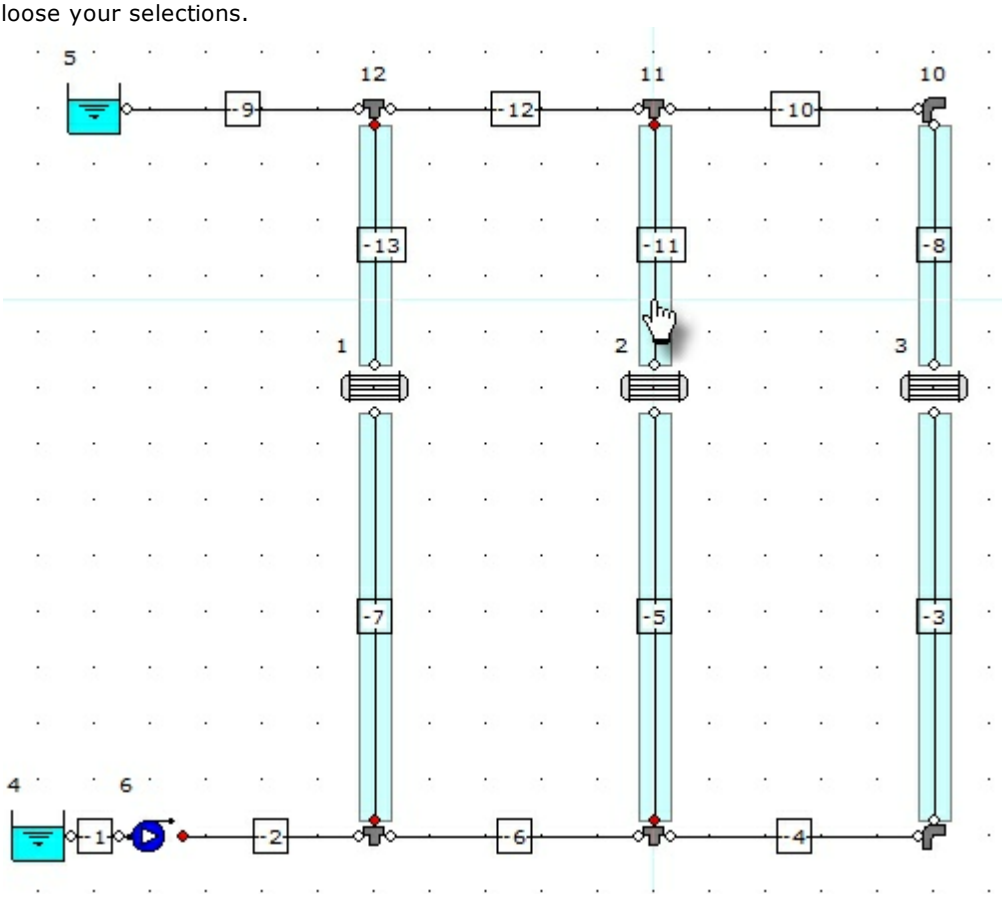
To enter the pipe lengths we can use one of two approaches. Either we can select pipes from the flowsheet or we can select pipes from the Lists Tab in the data palette. We will use the flowsheet in

© <2013> ... Flite Software NI Ltd

this example.

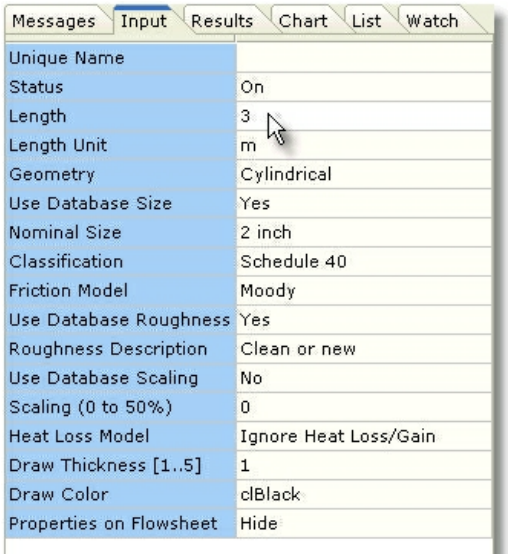
We can reduce the amount of data entry we make by recognizing the fact that some of the pipes are identical. For example the main feed and return branches to each exchanger are identical (pipes -3, -5, -7, -8, -11 and -13). If we use the fact that we can make multiple selections on the flowsheet we can change the length of all 6 pipes with one edit.

There are many ways to make multiple element selections, but for now we will use the mouse click method. To make multiple selections using mouse clicks on the flowsheet simply hold down the Shift key and click each element you wish to select. If you make a mistake and select the wrong element just click the element again and it will deselect. Don't forget to keep the Shift key depressed as you are making the multiple selections. Use this method to select the 6 identical branch pipes. If you release the shift key and click anywhere on the flowsheet other than on a selected element you will loose your selections.



To enter the pipe length of 3 m for each selected pipe, click on the Input tab in the data palette, move to the Length row in the Input Inspector and change the length to 3.

© <2013> ... Flite Software NI Ltd



<---- Input Inspector

The length of all 6 pipes is changed in one edit. Change the length of the remaining pipes, (Hint: the header and return sections -3, -5, -9 and -11 are identical).

Time to save our work. Use the File Save menu to save your work now. It is good practice to regularly save your work.

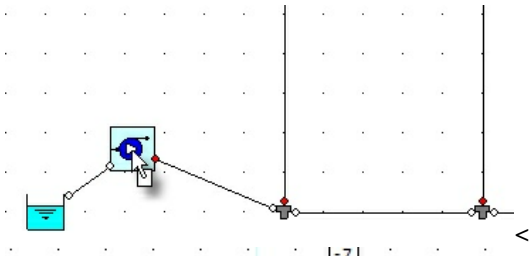
To complete the pipe data entry we need to make 2 additional entries. For each pipe we need to specify a nominal size and we need to add further bends as shown in Table 3. We need to determine pipe size and FluidFlow can help us here, so we will defer this task and add the additional bends now.

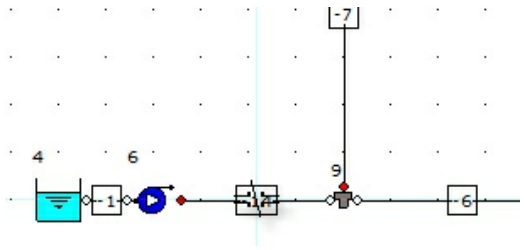
There are 2 additional bends in each branch line, 3 in the supply line from the known flow to the first branch (8) and 5 additional bends in the return line from the last return branch (11) to the cooling tower.

As we are dealing with an incompressible fluid, where the density change is small throughout the network we can avoid entering all bends individually. Instead use the Quantity row for each bend we add in the Input Inspector to reduce the number of bends we need to add.

Note: This approach is NOT recommended where density changes throughout a pipe section are significant.

Select the Junctions Tab in the component palette and click on the bend, then drop this bend into pipe. If you need to create some additional length to pipe -1 on the flowsheet, click on the pump node (6), hold down the left mouse button and drag the node to a different location.
Both of these operations are shown below.

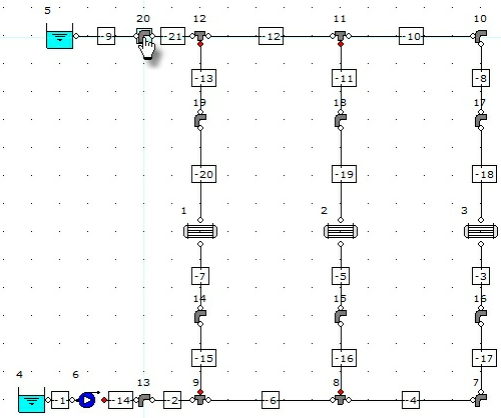

<---- Dragging a node



<---- Inserting a bend

Add the remaining bends as shown highlighted in the flowsheet below.

© <2013> ... Flite Software NI Ltd

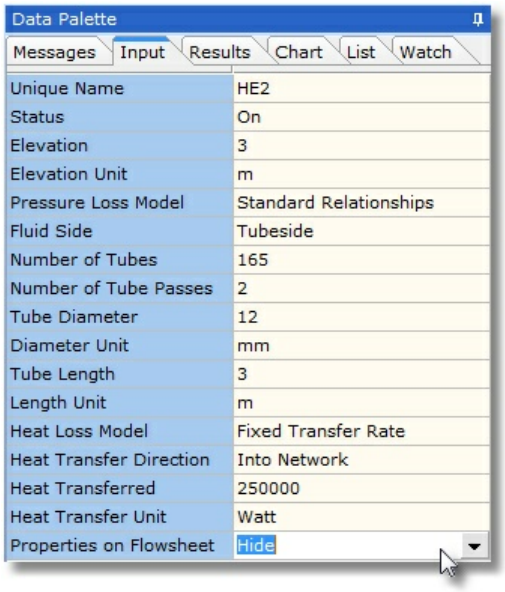


Hold the Shift key and click on bends 14, 15, 16, 17, 18, and 19 and change the Quantity row in the Input Inspector on the Input page of the data palette from 1 to 2.

Click on bend 13 and change the quantity to 3. Click on bend 20 and change the quantity to 5. This completes the data entry for the pipe data given in Table 3. All that remains for pipe entry is to set each pipe size.

You may have noticed that inserting an element in a pipe splits the pipe lengths into two equal sections, this is the default behavior but can be changed if desired.

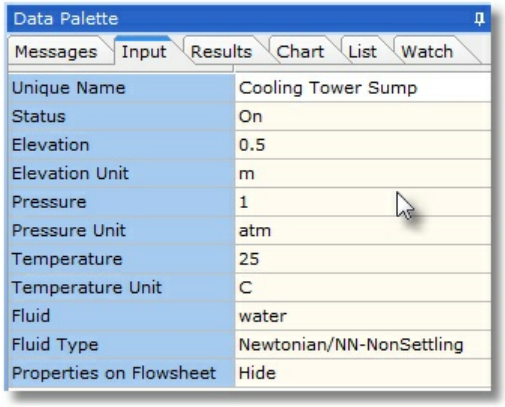
Click on each heat exchanger and change the default data to reflect the tube information provided in table 1. The Input Inspector for HE2 is shown below.



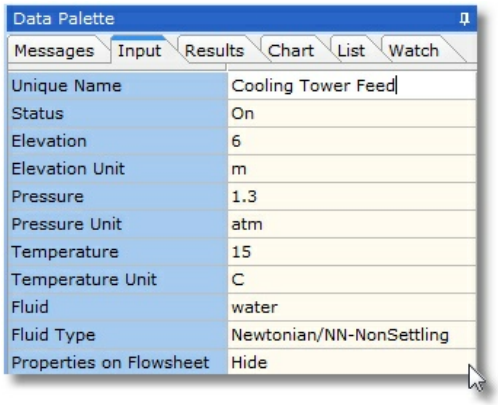
You have probably realised, that the number and content of the rows displayed in the Input Inspector is governed by the choices you make. For example changing the Heat Loss Model From Ignore to Fixed Transfer Rate means that you need to supply additional data and so the additional rows Heat Transfer Direction, Heat Transferred and Heat Transfer Unit appear. Heat Transfer Direction Into the network means that the exchangers are acting as coolers i.e. the process side is generating heat.

Two more entries in the Inspector and we are ready to make our first calculation. At the input boundary (4) we need to set the pressure, boundary temperature and ensure that the fluid is water. The Input Inspector should look like. (25°C inlet temperature and a pressure of 1 atm).

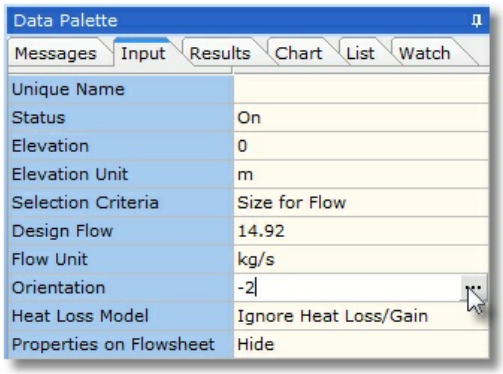
© <2013> ... Flite Software NI Ltd



In the known pressure outlet boundary (5) we need to set the pressure to 1 atm + 30000 Pascals needed to overcome losses in the cooling tower flow distributor. We can leave the temperature and fluid at the default entries since the flow will be out at this boundary and therefore the temperature, flow and fluid will be determined by the calculation.



At the auto pump (5) we need to enter the desired flow of 14.92 kg/s. The orientation of the pump should be set so that the red dot, which represents the pump discharge, points to pipe -2. If you need to change this click on the Orientation row in the input editor.



Before we make the first calculation you should also check that elevations are correct for each element. In fluid flow calculations the relative elevations are important, which means that we need to select a datum or grade point. i.e. a point where all elevations are measured relative to. Normally we would select the ground to represent a 0 elevation. In this example we will take the pump centreline as representing 0 elevation. Check that your node elevations are;

Node 4 at 0.5m
Node 6 at 0m
Nodes 13, 9, 8, 7 at 0m
Nodes 14, 15, 16 at 1.5m
Nodes 1, 2, 3 at 3m
Nodes 19, 18, 17 at 4.5m
Nodes 10, 11, 12, 20, 5 at 6m

To complete the data input make sure that all pipe lengths are set to the values in table 3. At this stage we can leave the pipe sizes at the default values because our next task is to size all of the pipes.

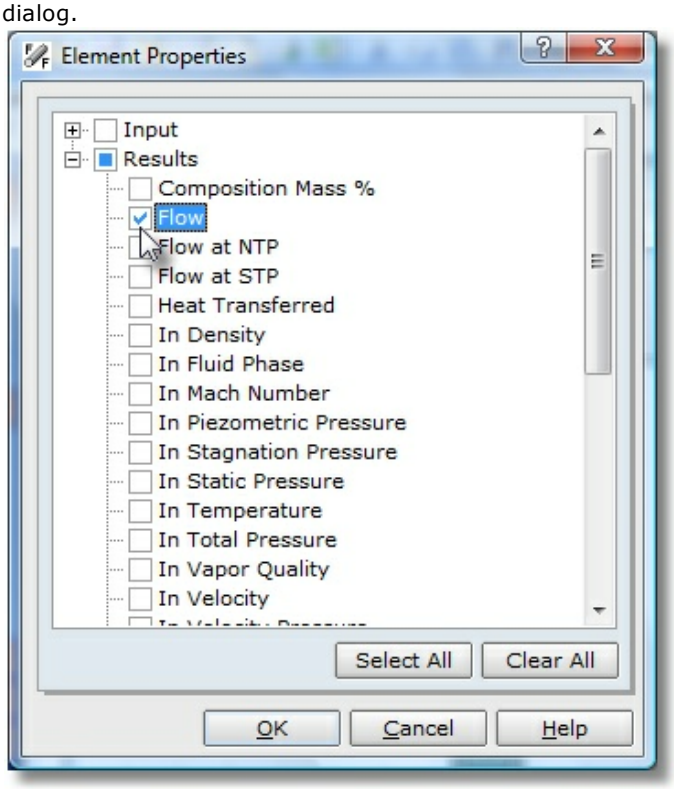
© <2013> ... Flite Software NI Ltd

If you wish that check that your data entry is correct, you can load up the example "\Examples\Cooling Water Distribution. First Design Iteration" which can be found in the Examples folder.

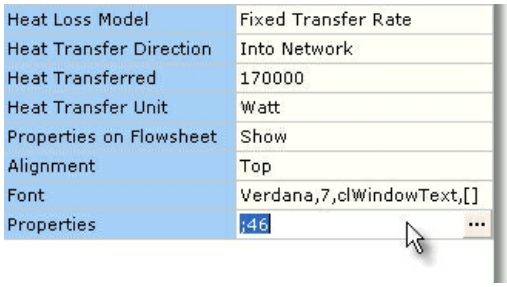
Press the calculate button, you should see FluidFlow quickly solve the network and flow directional arrows appear on the pipes in the flowsheet.

You can view the results in many ways. The simplest is to select the Results Tab in the data palette and click on each heat exchanger in turn on the flowsheet. In the results table the only row we are interested in at this stage is the mass flow through each exchanger. If we click on each exchanger in turn we can see that the flows do not match what we need from a cooling viewpoint. This means that the cooling system is unbalanced and is will not work as specified in the initial design definition. You should be able to see that the flow through HE1 is greater than design and the flow through HE2 and HE3 is too low.

There is a useful way to view these results. Since we will be constantly referring to these flows we can show them on the flowsheet. To do this click on the 3 exchangers, while holding down the shift key to make a multiselection then in the Input Inspector, change the Properties on Flowsheet row from Hide to Show, set the Alignment to Top and press the Properties button to obtain the following dialog.



Check the Flow in the Results tree and press the OK button. The rows in the Input Inspector should look like



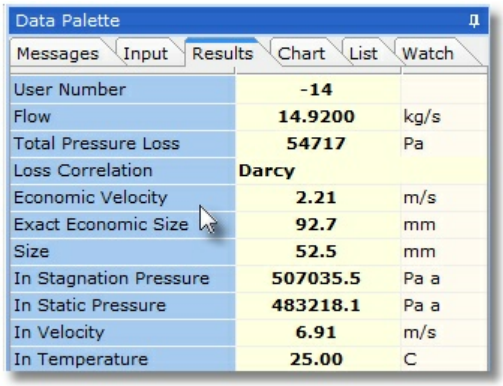
and the results will be shown on the flowsheet. If you wish to cross check your calculation at this stage you can load up the file "\Examples\Cooling Water Distribution. Second Design Iteration"

Pipe Sizing:

Before we balance the network we need to set the pipe sizes. In FluidFlow each time the pressure loss through a pipe is calculated, its economic velocity and hence the economic pipe size are also

© <2013> ... Flite Software NI Ltd

calculated. By default these values are shown in the results table. Select the Results Tab in the data palette and click on the branch pipes in the flowsheet. As we move from each branch pipe we can see that the Exact Economic Size row in the results table will be around 50mm. This means that we should use the nearest standard pipe size 50mm (2") pipe in the branches. Since the default pipe size is 2", (unless this has been changed or you are using a different environment) which means we do not need do make any size changes to the branch pipes. Use multi select in the flowsheet to select all the branch pipes and change the size if you need to do this.



<--- Economic pipe size

Click on the supply or return header pipework and we can see that the size needed ranges from 93mm to 50mm. There is a case for reducing the header size after each take off. This will may reduce costs, since we can utilise reducing Tee's.

We will use the economic size suggestions to change the pipe sizes in the following manner.

Set the pipe size in the supply header to the first branch (9) and in the return header from the last branch (12) to be 4" (Pipes -2, -14, -21 and -9).
Set the pipe in each header between the first and second branches to be 3". We do not need to make adjustments to the rest of the header as it is already at size 2".

Recalculate and save your work at this point. You may wish to load up the example "\Examples\Cooling Water Distribution. Third Design Iteration" which can be found in the Examples folder, so that you can check that your results are similar.

FluidFlow calculates correctly for reducing Tees, provided that you are using Idelchik, Miller or SAE types (this is the default). The tee's at nodes 8 and 11 have connecting pipe sizes 4, 3 and 2" and pressure conversion effects from velocity to static or vice versa are taken into account when calculating pressure losses at the tee. Click on the tee, select the Input Tab and click the Nomenclature row in the Input Inspector if you need further information. For tees having 3 different branch sizes the loss relationships need to be extrapolated and you may find that you have warning messages to this effect.

Warning messages are there to help you decide if you need to make design changes. In this case the warning messages refer to the possible loss of calculation accuracy in the tee junctions because relationship data has been extrapolated. Since there are no other available pressure loss relationships available for these types of reducing tees we have no choice but to accept this warning. Still it is worthwhile checking on the calculated K values to ensure these are within an expected range (-2 to 10). You can also cross-check by using another loss relationship (say Miller type) and verify that the calculated K values and pressure losses are similar. This is the case here and so we can safety ignore the warnings. In fact we can turn off some of the less severe warnings, but this is not recommended.

Often, as engineers we like to keep header and return line sizes equal along the header and so a 4" or even 3" header/return line size is also a valid solution. Remember to take into account all possible operating scenarios and future considerations before making your final design decisions. For example, if we knew there was a possibility of a 4th heat exchanger being added at some time in the future then it would be a better solution to make the header and supply lines all 4". Pipe line sizing is always a balance between capital costs, operating costs and operating flexibility.

Pump suction lines should always be given careful consideration. We must always ensure that we have an entry head at the pump suction above the net positive suction head required by the pump + a safety margin. FluidFlow will detect and warn if adverse conditions exist and as a first guess we will use 4" pipe.

Balancing the network:

© <2013> ... Flite Software NI Ltd

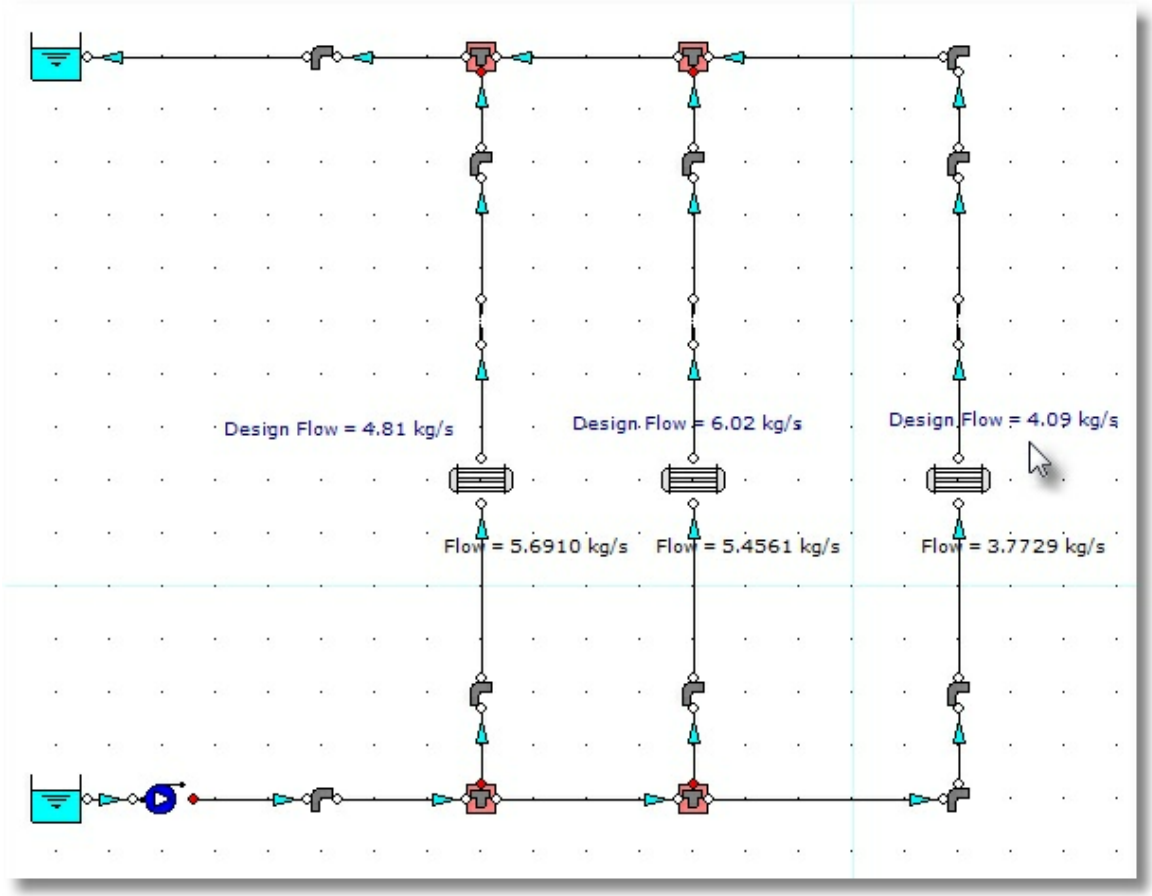
We need to add additional elements in order to balance the network. Of course we can achieve the balance directly if we use a flow controller in each branch, and this may be a valid solution.

In this example we will introduce orifice plates to drop pressure in a controlled manner in each branch so that the correct flow balance is achieved. Balancing is wasteful of energy, however we achieve our distribution goal.

To obtain the distribution required we will use 3 orifice plates, one in each branch. Using orifice plates is a cheap solution to the distribution issue. However it may not prove very flexible if process conditions are likely to change. In this case using throttling valves may be a better solution

If you have the autoselect module this can be accomplished automatically. Without this module we need to use trial and error.

Add the orifice as shown in the flowsheet, set the Elevation of each orifice to be 3.75 m and the diameter to be slightly below the default pipe size say 50 mm. Recalculate and we can see a different flow distribution.



We have added some free text to the flowsheet to indicate the design flows that we are trying to achieve. Use the Text tool on the flowsheet palette to do this.

In our trial and error approach to flow balancing we will use the following technique.

1. Reduce the size of the first orifice (21) until we reach a flow slightly below the design flow. (after a few iterations you will arrive at a size of 34 mm). Note the flow balance is almost acceptable.
2. Reduce the orifice size in the third branch (23), until we obtain the slightly below the design flow. You should end up with a size of 43 mm.
3. Adjust the size of the middle orifice until the overall balance is to the required flow (+- 5%). You should end up with a size of 47mm

The final result is stored as "Cooling Water Distribution. Final Calculation". You should also note that it is quite possible to obtain different orifice sizes than those provided above and still obtain the balance required.

We have now almost completed the design. We will look at how to make final equipment

© <2013> ... Flite Software NI Ltd

specifications and how to consider the effect of likely operating scenarios in Part 2.

Summarizing you should have learned the following skills:

- How to select components from the component palette and how to place and move them on the flowsheet.
- How to connect pipes between nodes.
- How the Input Inspector works.
- How to make data changes to the flowsheet elements, both individually and as a selected group.
- How to show result text on the flowsheet
- How to interpret the calculated results to select or optimise a pipe size.
- How to balance a fluid network using orifice plates.

© <2013> ... Flite Software NI Ltd

1.10 Design of a Tank Farm Gas Collection System

In this example we are not concerned with building the model and data entry. Instead we focus on the engineering.

Problem Statement:

It is desired to collect together the air vents from a group of 5 storage tanks holding a flammable, obnoxious liquid. The vent gas is to be treated in an activated carbon bed before finally passing to atmosphere.

In this problem we are only concerned with the situation that occurs as the tanks are filling. In our scenario each tank, vents gas (for simplicity considered to be air), at the rate of which it can be filled. The exit gas vent from each yank is modeled as a known flow element, with the maximum tank fill rates already entered.

The remainder of the collection network has been built and can be opened from the \Examples folder "Non Sized Gas Collection System". Open the example and consider how we may answer the following question.

1. Size all pipes so that the operating pressure under maximum filling rates does not exceed that allowed under code API 650. This means no more than 1 psig operating pressure in each storage tank.

Considerations and approach:

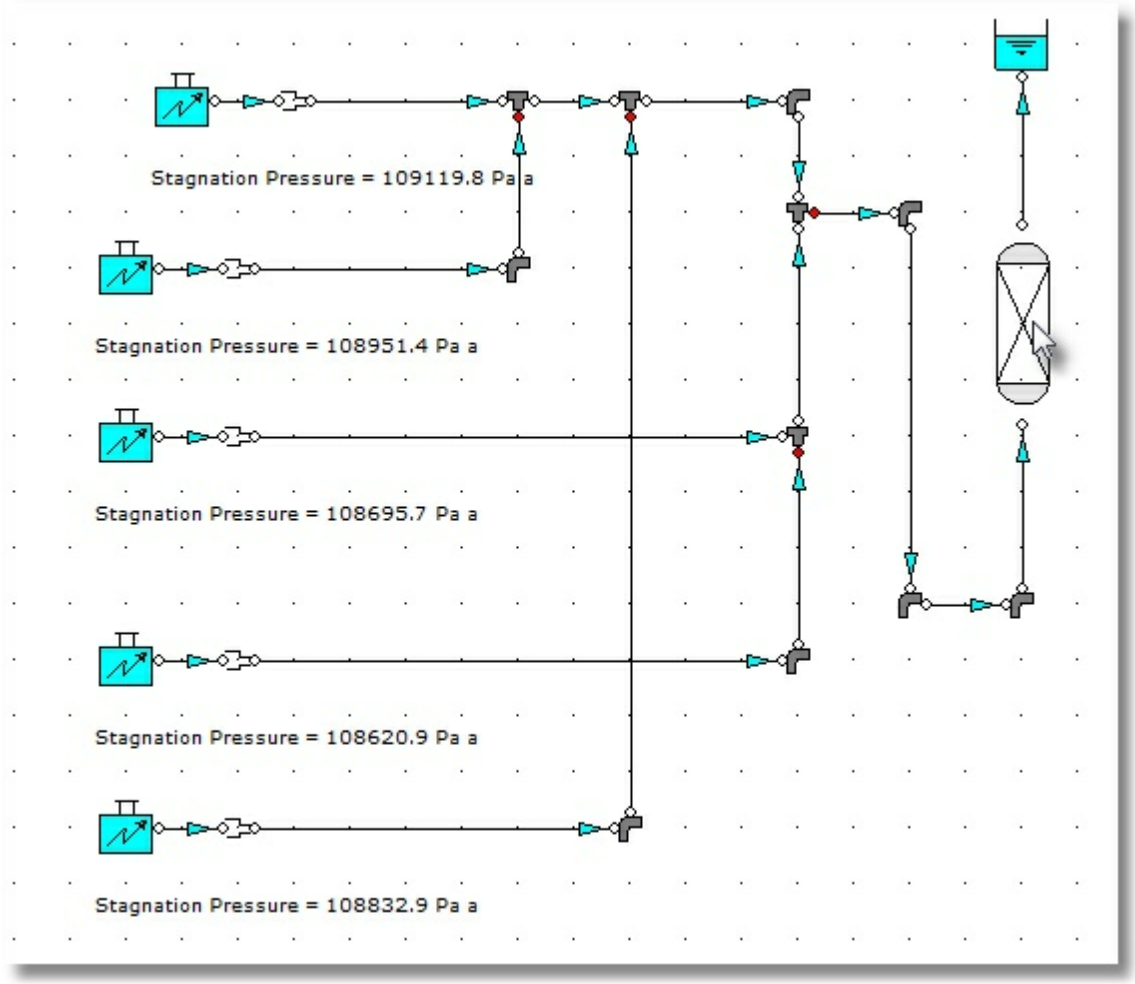
We will take the worst case of all 5 tanks filling at one time.

To easily view the calculation result values we can use one of three possible techniques:

- We can configure and turn on the fly by results. In this way we can move the mouse over the flowsheet and look at the result values we are interested in.
- We can click the Results tab in the data palette on the left click on the flowsheet to select each component we are interested in.
- We can show the results on the flowsheet.

It is important to state at the outset that there are many solutions to this problem. For example we could increase all line sizes until the pressure in each tank dropped below 1 psig. This would undoubtedly work, but as the lines are made of stainless steel and are of a reasonable length we may not wish to over design in this way due to cost considerations.

© <2013> ... Flite Software NI Ltd


Stagnation Pressure = 109119.8 Pa a
Stagnation Pressure = 108951.4 Pa a
Stagnation Pressure = 108695.7 Pa a
Stagnation Pressure = 108620.9 Pa a
Stagnation Pressure = 108832.9 Pa a

To start, let us consider the tank pressures if we use 2" pipe throughout (this is the case if we open the example). You can see that each tank is over the maximum allowed pressure of 1 psig.

Click on any pipe and look at the results table in the data palette. You can see immediately that each pipe is already larger than the size recommended according to the economic size. This is an important point and illustrates that you cannot blindly set all pipe sizes to the suggested economic size in all cases.

Click on a few more components and consider the row titled "Non Recoverable Loss", this loss represents the pressure loss that can never be recovered. It is this value that we must impact (reduce) if we are to design a safe system.

You should quickly note that the majority of the system pressure loss occurs at over the packed bed. The packed bed represents a pressure loss of 0.6 psi out of a total of < 1psi available.

Using this knowledge, perhaps the best approach to take is increase the diameter of the bed to reduce the pressure loss, rather than increase the pipe size of each pipe. This now becomes a cost issue. For example is it less expensive to change the diameter of the bed, or use a different particle size in the bed rather than change the pipe sizes.

We do not have sufficient information to fully consider the available choices. What is important is that you recognise how to ustilise the power of FluidFlow to consider the alternative design scenarios.

If you have the scripting module you can automate this process.

An acceptable design changing only pipe sizes downstream of the Tee junction (node 15) is saved as "Pipe Sized Gas Collection System".

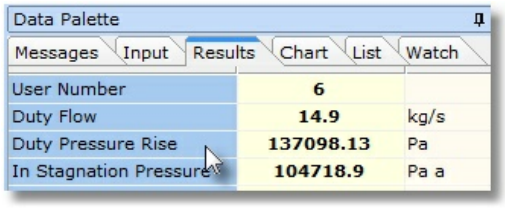
© <2013> ... Flite Software NI Ltd

1.11 Design of a Cooling Water System. Part 2.

This is the concluding part to the design of a cooling water system, started in Part 1.

In this part we will select a pump to perform at the operating duty specified from the auto booster element and consider the effect on the system if the heat load on exchanger 2 (HE2) is increased by 33%.

Open the file "\Examples\Cooling Water Distribution. Final Design Iteration" as the starting point for this exercise and then click on the auto booster element on the flowsheet and the results tab on the data palette. If you are using the default "System International" environment you should see a result similar to;

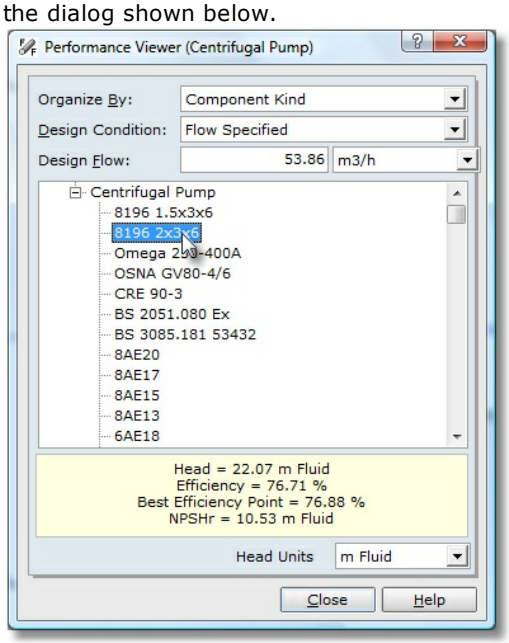


i.e. the design flow of 14.92 kg/s of water at 25° C needs a pressure rise of 137098 Pa in order to operate at the design flows. We may wish to stop here and ask our preferred pump manufacturer to suggest a pump to supply this duty. To do this we will need to convert the pump duty flow to a suitable volumetric units and the calculated pressure rise needed to head units. You can use FluidFlow to do this for you automatically, however you may find it instructional to do the conversion now to m3/h and m Fluid. The density of water, obtained from the results table is 997 kg/m3 and so the calculated volumetric flow will be 14.92 x 3600/997 = 53.87 m3/h. The head that the pump is required to produce will be 137098/(997 x 9.80665) = 14.02 m Fluid. This information, together with the operating fluid and temperature conditions is enough for the pump manufacturer to make a selection and under normal circumstances this is all that is required.

It is better to allow the manufacturer to make the selection for the following reasons; pump selection is often more than a simple hydraulic selection. Pump configuration, sealing and shaft load considerations, materials of construction etc are best handled by the manufacturer.

Even though the manufacturer is in a better position to make the selection it is still worthwhile and you can often get a more flexible design by making some additional hydraulic considerations.

For the purpose of illustration we will use the inbuilt pump selector to make the selection at this duty point.
As we wish to select a pump without the auto select module, we aid our task by using the following tool. From the menu select 'Tools | Equipment Performance Viewers | Pump Performance' to create the dialog shown below.



To use this tool change the Design Flow to be 53.86 m3/h, and as you click on each pump in the list the performance data (Head, Efficiency, Best Efficiency and NPSH required) are shown for the currently highlighted pump.

© <2013> ... Flite Software NI Ltd

This means that you can move through each pump in the database and view how it will perform in this system. The pump shown above would be a suitable selection from NPSH and efficiency viewpoint, but is unsuitable because the head developed is significantly greater that that required by our duty (14.02m).

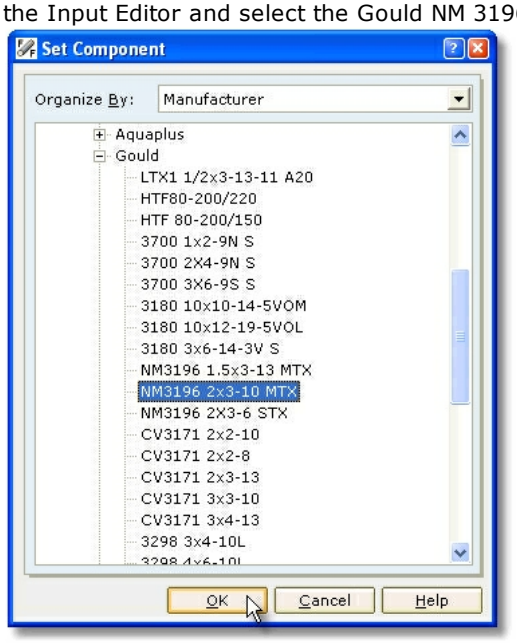
Viewing the pump database provides a number of pumps that will fulfill the hydraulic and system needs. You may wish to consider some of the following models: Girdelstone 32ns; DNP85-165; FA 253-4402Z; NM3196 2x3-10 MTX etc.

The best choice is the to select a pump operating near the best efficiency or a pump with the highest efficiency at the duty point.

To complete the example we will select he Gould pump NM 3196 2x3-10 MTX, even though this will provide approx 20% more head than required at the duty flow.

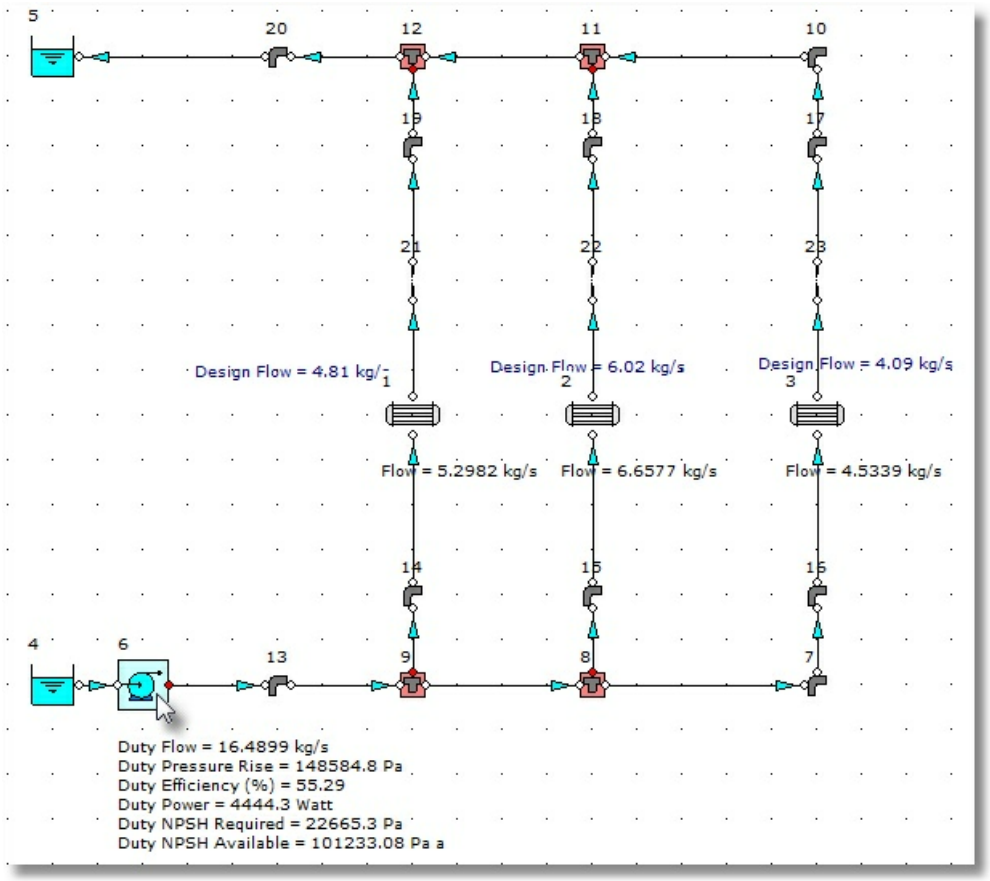
To achieve this right mouse click on the auto pump node (6) and select Change Component from the pop up menu, change the element type from auto booster to centrifugal pump. Click on the newly created centrifugal pump and from the Input Editor click on the orientation row so that the pump discharge side (denoted by the red dot) is oriented in the correct direction.

Change the pump model from the default pump, by clicking on the button in the Pump Model row in the Input Editor and select the Gould NM 3196 2x3-10 MTX



Notice that the pressure developed by this pump is around 20% more than needed and that the flows through each exchanger have increased by around 10% above the design flows.

© <2013> ... Flite Software NI Ltd



As engineers we like to over design. Right !!!. Before we congratulate ourselves on the design we may wish to consider that the selected pump is operating very near to the run out condition. This is wasteful of energy, has a larger NPSH requirement, increases pump wear and we would need to oversize the motor to operate at this condition.
We should select a smaller pump, or better still we can reduce the impeller diameter to from 254mm to around 239mm. This will allow us some additional capacity if needed in the future and enables us to more closely meet the design conditions (< 2%).

The system calculation using this pump with a speed reduction to 1360 instead of a reduced impeller size, is saved as "Cooling Water Distribution. Final Design Iteration. Pump Selected and Adjusted".

WARNING
You should note that reducing the impeller diameter or changing the speed will produce a flexible design, but in this example, you can see from the adjusted pump chart that we are still operating away from the best efficiency point and so this may not be the best solution.

Finally we need to consider increasing the heat load to HE2 by 33%. Change the heat transferred to be 333,000 Watt, then recalculate. The temperature ex the heat exchanger rises from 34.9 °C to 38.2 °C. The temperature to the cooling tower increases from 34.9 to 36.2 °C.

This increase in temperature is considered to be acceptable and we do not need to rebalance the system.

© <2013> ... Flite Software NI Ltd

1.12 Configuration and Environment

Nearly all aspects of the FluidFlow application can be configured and customized. The configuration settings that you change are saved, so that each time the application starts your own preferences are applied. Some of the configurations will be familiar, such as changing the position of tool bars, adding buttons, etc., because these customizations are commonly found in professional software.

Some aspects of the environment are stored as a subset called an Environment Set. You can make as many environment sets as you need and change between them interactively.

An environment set is closely associated with the default input data and units for every component (fluid equipment item) that is available from the component palette as well as how and what calculation results you wish to see.

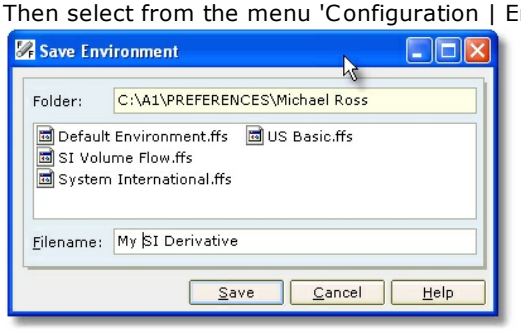
Each environment set stores the following information, and can be easily accessed via a function key:

- F4 - Provides access to the default settings for each component available from within the program.
- F5 - Provides access to the data columns you wish to export to Excel.
- F6 - Allows you to set up Fly By Options. A 'fly by' is the window that appears as you move the mouse over a component on the active flowsheet. It is possible to set the fly by content for each component.
- F7 - Provides access to the columns you wish to print in your report, or export to Word, HTML, or PDF.
- F8 - Allows you to configure the contents of the table shown on the Results tab of the data palette.
- F9 - Allows you to individually set the calculated result units and the number of decimal places you wish to use.

After a new installation of FluidFlow there are 2 environment sets already made for you. "System International" and "US Basic". These should form the basis of changes you make. Rather than making the changes to any of the basic sets provided, it is a better idea to make a renamed copy and make the changes to the copy.

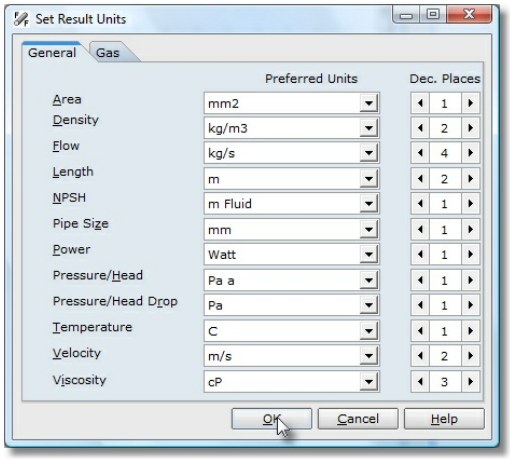
As an example of how to do this. First change environment to that you wish to copy from, you can do this from the combo box on the main application toolbar.

Then select from the menu 'Configuration | Environment | Save Environment' to a new name



Use the combo box in the main toolbar to change to the newly created environment and then customize by say pressing F9 to change the result units in this set.

© <2013> ... Flite Software NI Ltd



Here we have made the following changes from the original copy, we are displaying flow in l/min instead of kg/s and we are displaying both pressure and pressure drop in bar a and bar. The number of decimal places shown has also been increased to reflect this change.

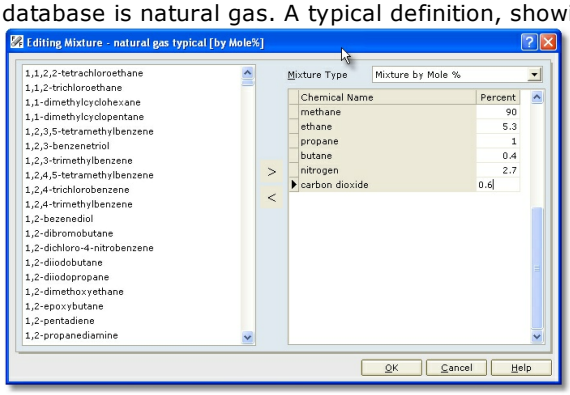
Load up an example such as "QA Incompressible Flow\Boosters\4 Pumps in Parallel 3 Operating" which has text on the flowsheet. Move to the Results tab on the data palette and then change environment set. You will see that the flowsheet results and the table results update immediately to show the new conditions.

1.13 Databases

A powerful feature of FluidFlow are the many associated databases that support and enhance the application. These include a database of fluid properties, databases that describe the performance and limits of fluid equipment items, materials databases that hold pipe sizing and pipe insulation information, and a database of manufacturers, costs, and user-defined areas of application.

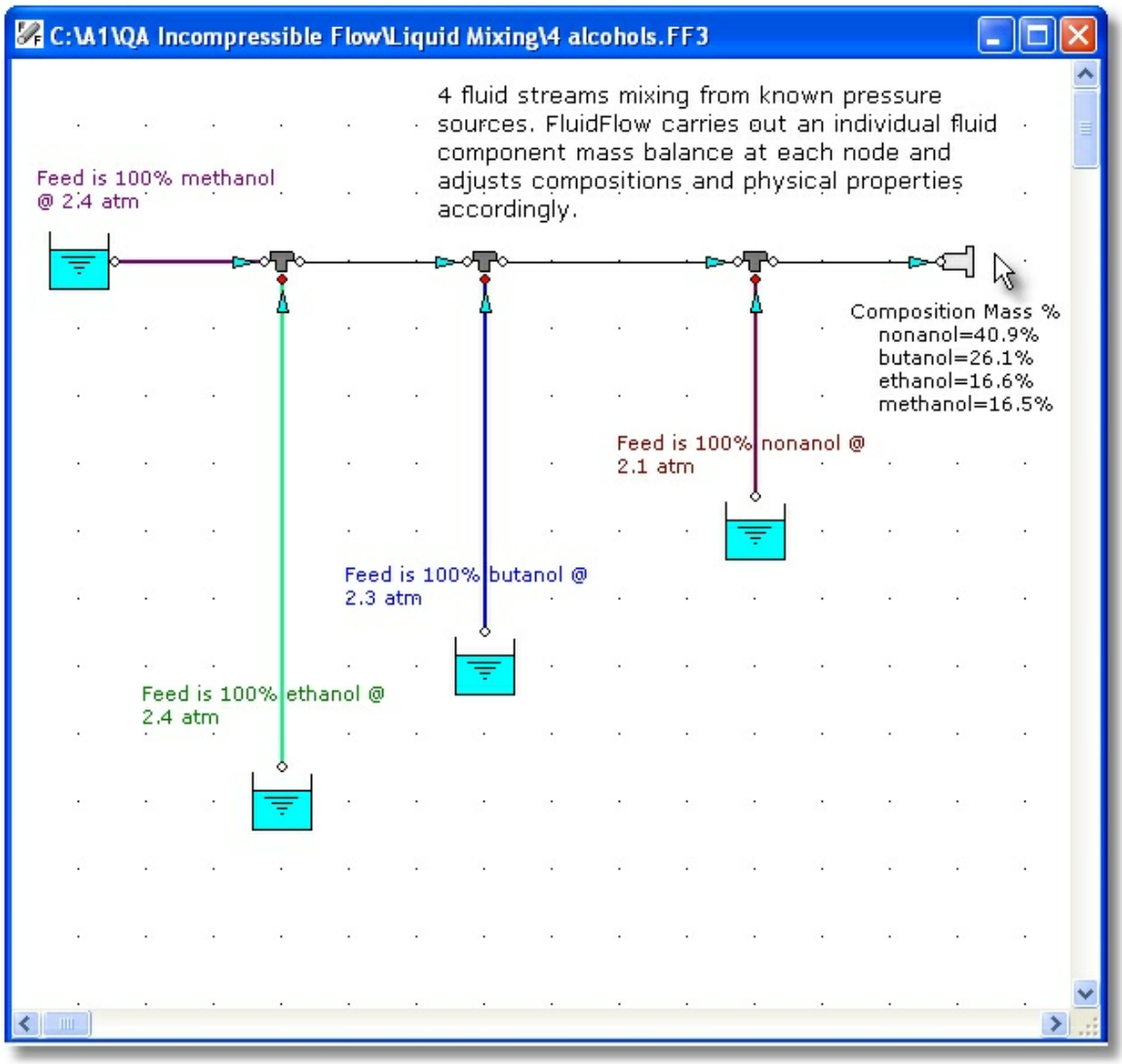
1.14 Fluids Database

The fluids database contains comprehensive thermo physical data for over 850 fluids. Thermo physical properties (density, viscosity, thermal conductivity, specific heat, physical constants and critical values, heat of vaporization, vapor pressure, and surface tension) are stored so that FluidFlow can complete pressure loss calculations including heat transfer and phase change.
The database is much more than a table of physical properties. Many "state of the art" physical property prediction methods are available, often used together with modern Equations of State such as Benedict Webb Rubin, Lee Kesler, and Peng Robinson. There are special relationships for water and steam (IAPWS), air, and you can also mix fluids (non-reacting) by using the database tools or dynamically in the flowsheet.
An example of a pre-mixed fluid made by combining fundamental fluid components in the fluid database is natural gas. A typical definition, showing molar composition is illustrated below:



Alternatively fluids can be mixed on the flowsheet. The following example shows the mixing of 4 alcohols.

© <2013> ... Flite Software NI Ltd


C:\A1\QA Incompressible Flow\Liquid Mixing\4 alcohols.FF3
4 fluid streams mixing from known pressure sources. FluidFlow carries out an individual fluid component mass balance at each node and adjusts compositions and physical properties accordingly.
Feed is 100% methanol @ 2.4 atm
Composition Mass %
nonanol=40.9%
butanol=26.1%
ethanol=16.6%
methanol=16.5%
Feed is 100% nonanol @ 2.1 atm
Feed is 100% butanol @ 2.3 atm
Feed is 100% ethanol @ 2.4 atm

The scope of explaining the addition of a new fluid to the fluid properties database is a little outside the quick start guide. For detailed information on how to do this see the help file or user manual.

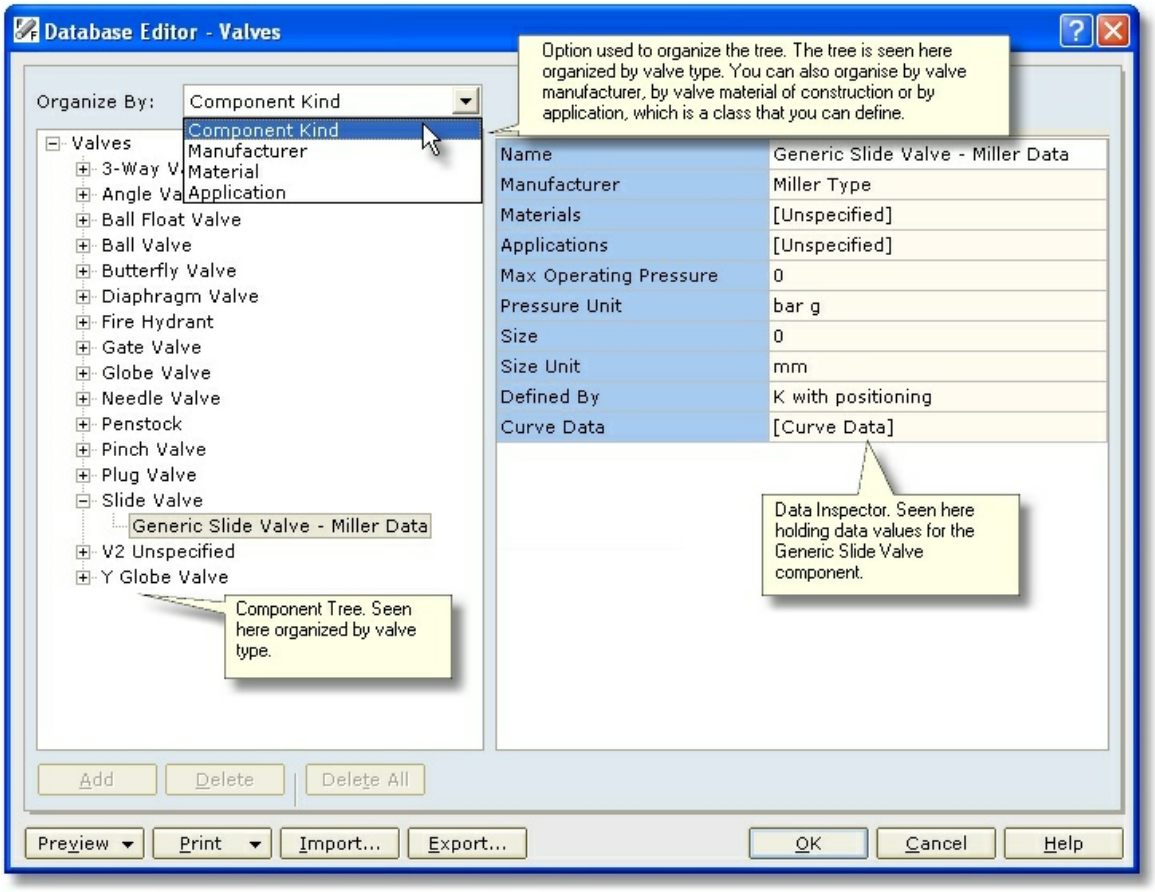
In the database section we will explain a simpler addition in the next section.

© <2013> ... Flite Software NI Ltd

1.15 Database of Manually Operated Valves

As an example of how the various databases are structured we will look closer at the dataset of manually operated valves. The dataset of manual valves is actually a subset of a larger database containing all fluid equipment components or items. The full database describes performance, limitations, and usage of all items on the component palette.

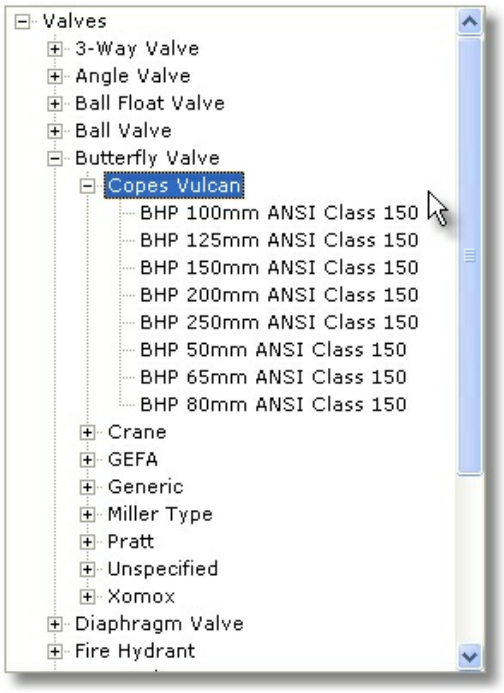
To view, edit, delete, or add to the manual valves subset, make the selection from the 'Database | Valves' menu, to display the 'Database Editor - Valves' dialog:



The Database Editor is also used for viewing, editing, deleting, and adding other fluid equipment items so the skills we learn here can also be applied to other items of fluid equipment.

On the left side of the editor dialog is a tree control that enables you to quickly access the individual item you need in the hierarchy. You can reorganize the tree if this is more convenient. Below we see the valves data tree re-organized according to manufacturer. In this dataset there are 8 manufacturers of butterfly valves shown in alphanumeric order.

© <2013> ... Flite Software NI Ltd



Adding new Data:

With all FluidFlow datasets the method of data entry is identical. In general we follow these steps:

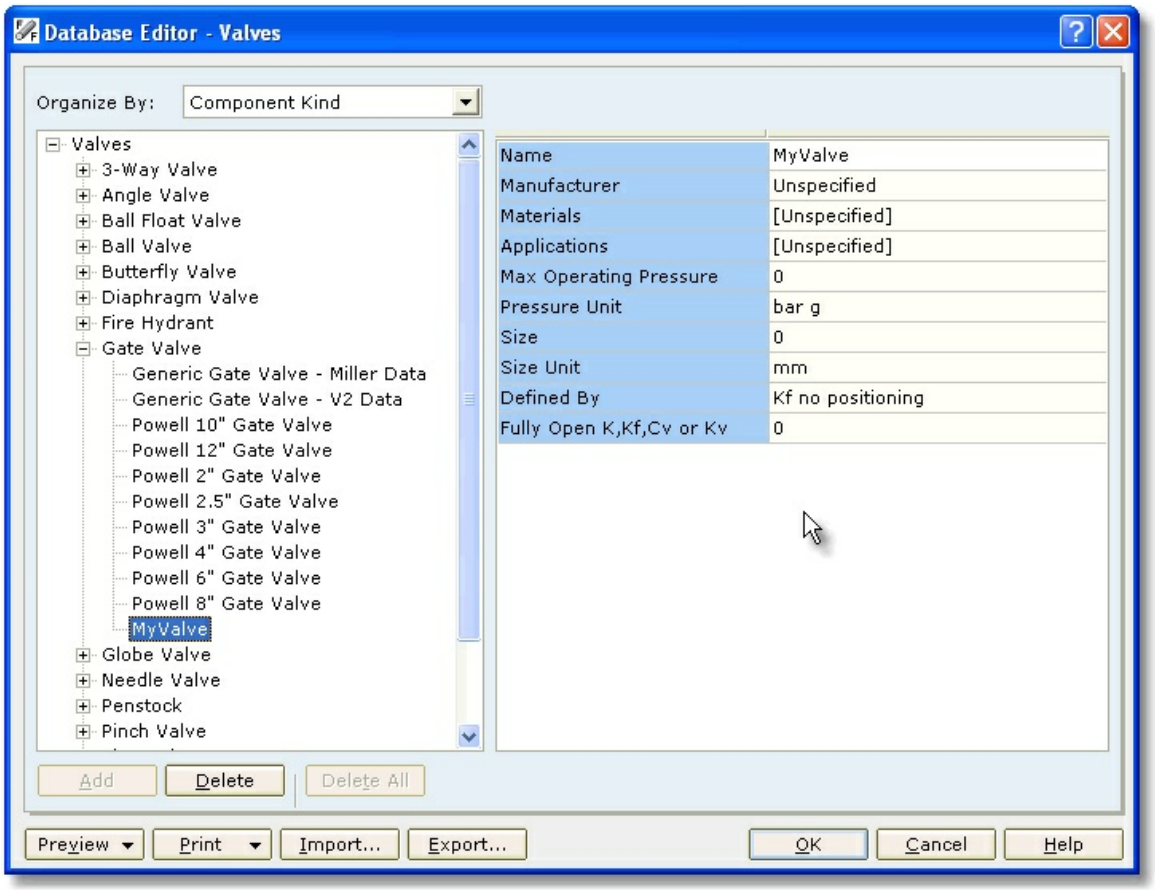
1. Select the appropriate dataset - use 'Database | Dataset Name' from the application menu.
2. Press the 'Add' button.
3. Enter a unique name and press 'OK'.
4. Enter the data that describes the component (fluid equipment item).

Add a new Valve:

To add a new gate type valve, select 'Database | Valves' from the application menu. This will display the 'Database Editor - Valves' dialog, identical to that shown earlier in this section. Organize the tree according to Component Kind and click on the branch titled "Gate Valve".

After clicking at this branch the 'Add' button should be enabled. Click the 'Add' button enter a name say "MyValve" and press the 'OK' button. The Editor should look like:

© <2013> ... Flite Software NI Ltd



You can now edit the default data via the Data Inspector. For example you could assign a manufacturer or several materials. The data available for this valve is provided by the manufacturer:

Size = 100mm with pressure loss data expressed as a Cv value in usgpm/psi. The pressure loss data is tabulated in Table 4

Table 4

% Open	Cv usgpm/psi
100	600
80	500
60	400
40	280
20	150
10	70

After you enter at least 3 sets of data points a curve fit of the data points occurs automatically. You can change the type of curve fit and if you select a polynomial you can also fix the order. The graph allows you to visually judge how the curve fit appears over the whole of the possible range. You can also zoom the graph for more detail and make a printout from this dialog.

We cannot emphasize enough, that it is vital that you check your data after adding any new component. You need to check the data for continuity over all possible operating positions/range and that you have also entered all additional data needed.

© <2013> ... Flite Software NI Ltd

1.16 Add a New Pump

As an additional exercise add a new (fictitious) centrifugal pump from the following data:

Max Operating Pressure = 10 m Water g
Suction Size = 150 mm
Discharge Size = 100 mm
Data Operating Speed = 100
Min Speed = 50
Max Speed = 150
Data Impeller diameter = 250 mm
Min Impeller Diameter = 200 mm
Max Impeller Diameter = 300 mm

It is possible to leave the suction and discharge values at 0. If you do this FluidFlow will not make any size check. FluidFlow makes size checks to ensure you connect up a correct pipe size to the suction and discharge flanges of the pump.

The Data Operating Speed and the Min and Max Speed can also be at the same value. You can do this if you do not intend to install a variable speed motor.

The Data Impeller Diameter and the Min Impeller and Max Impeller Diameters can also be set to the same value. This may be true if the pump only takes a fixed size impeller.

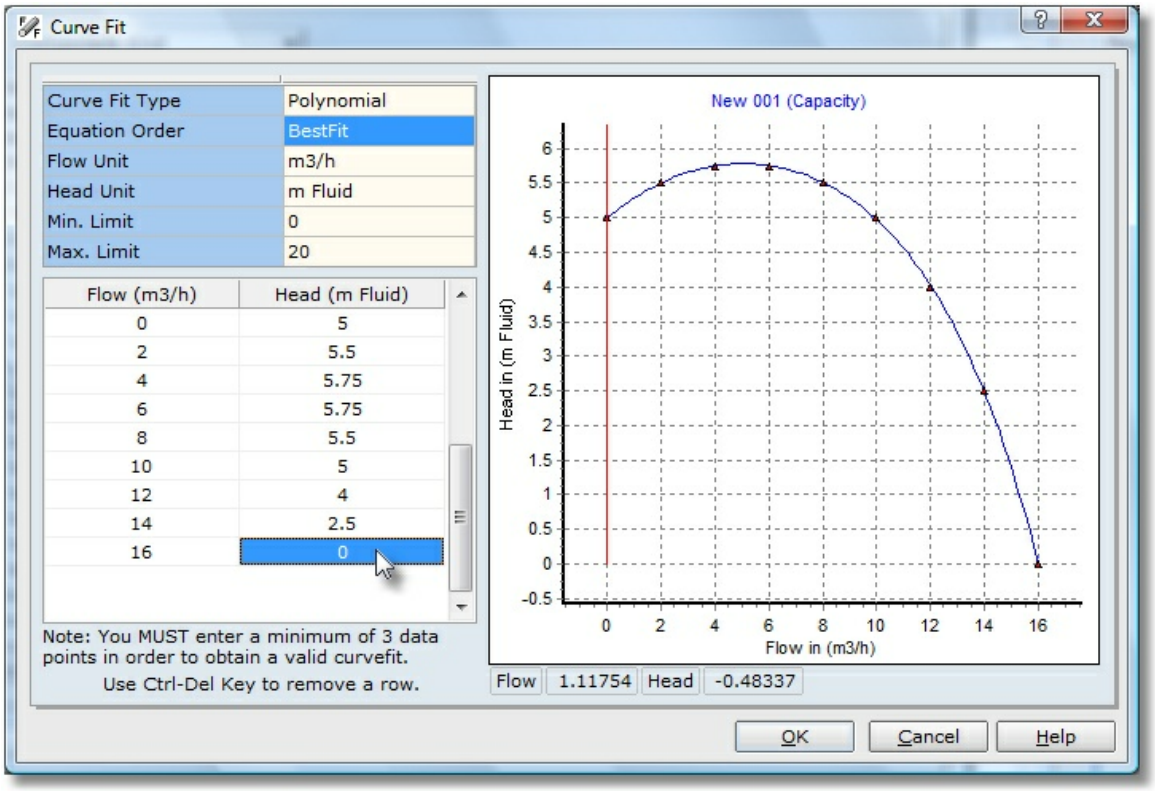
Then enter the capacity, efficiency, and NPSHr curves from Table 5

Table 5

Flow in l/s	Head in m Fluid	Eff (%)	NPSH in m Fluid
0	5.0	0	-
2	5.5	30	-
4	5.75	45	2.5
6	5.75	60	2
8	5.5	68	2.5
10	5.0	70	3
12	4	65	3.5
14	2.5	50	4
16	0	0	4.8

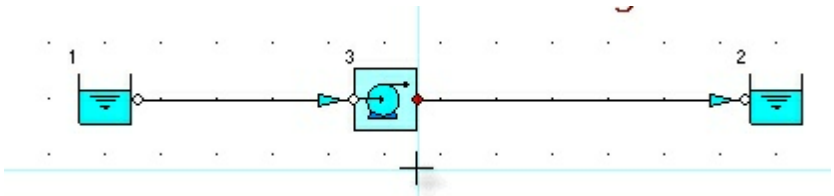
Min Flow limit = 2 l/s
Max Flow limit (run out) = 12 l/s
Name = [your own choice]

© <2013> ... Flite Software NI Ltd



An Important point is you should note is the entry at Head = 0. You are unlikely to get this information from any manufacturer, so you must make an estimate. The estimate you make can be guessed (this is OK because we will never operate the pump anywhere near to this condition). When you make the guess at head of zero, ensure that you do not "upset" the shape of the curve for the valid data points by selecting a value that lies on a smooth curve. If you do not make this guess you run the risk of using the pump in a system that is difficult to solve (converge).

Now that we have added the data save by pressing the 'OK' button and use the pump in the following simple flowsheet.



Check that the flow and head of the pump used in the system lies on the curve that you have previously entered.

© <2013> ... Flite Software NI Ltd

Index

- A -

Activation 1
activation code 1
Add a new pump 38
Add a new Valve 35
Adding new Data 35
Administrator 11

- B -

balance the network 14
Balancing 14
boundary 14
building the model 14

- C -

calculate button 14
component palette 13
Configuration 32

- D -

data entry 14
data input 13
data palette 13, 27
Database 35
Databases 33
default administrator password 11
default data 14

- E -

economic pipe size 14
economic velocity 14
Environment 14, 32
environment sets 32

- F -

flow balancing 14
Flowsheet 14
fluids database 33

- G -

gas collection 27

- H -

heat exchanger 14
Help 6
How to enter data 14
How to interpret results 14
How to make changes 14

- I -

Inlet Boundary 14
Installation 1, 6

- K -

Keeping current 5
known flow boundary 14
known pressure outlet boundary 14

- L -

Logon 11

- M -

model building basics 14
modules 1
multiple element selections 14

- N -

Network 6
network module 9

- O -

orifice plates 14
Outlet Boundary 14

- P -

physical properties 33
physical property prediction methods 33

© <2013> ... Flite Software NI Ltd

pipe length 14
pipe numbering 14
pipe size 27
Pipe sizes 14
Pipe Sizing 14
piping network 13
Product Id 1
pump 14, 29

- Q -

Quantity 14
quick key 32

- S -

schematic layout 13
Setting Up Client 6
settings 32
Setup 1
synchronised 13
System Requirements 1

- W -

Welcome 1

© <2013> ... Flite Software NI Ltd

Back Cover